A CFD study of the losses in sharp and radiused orifices with and without inlet cross-flow

Degree Project in Fluid Mechanics, Second Cycle
SG203X

Author: Tien Nguyen
901212-3015
tien@kth.se

Supervisors: Mihai Mihaescu, Royal Institute of Technology
Sara Rabal, Siemens Industrial Turbomachinery

Examiner: Anders Dahlkild, Royal Institute of Technology

17-11-2014
Abstract
Increasing efficiency demands on modern gas turbines require higher operating temperatures which increase the thermal load on the materials. The secondary air system transfers heat away from the components to ensure that the operating conditions are under the components’ critical values. In the secondary air system, orifices are an essential component, thus a thorough understanding of the flow behaviour, more specifically the mass flow and pressure loss, is crucial. Siemens Industrial Turbomachinery initiated an experimental study regarding the discharge coefficient for sharp and chamfered orifices (Binder, 2013). This work seeks out to enhance the previous study with radiused orifices through the use of Computational Fluid Dynamics (CFD). Additionally, inlet cross-flow was studied for radiused orifices.

A literature study was conducted to provide a deeper understanding of the flow behaviour and to provide a basis for comparison. A mesh verification study was performed to ensure that the meshes used were adequate, this was done only for flow cases without inlet cross-flow. A validation study was conducted with the data from Binder (2013) and Hüning (2012) for two representative cases without inlet cross-flow and one case with inlet cross-flow. The CFD calculations showed in general a good agreement with the previous work. Furthermore, a small turbulence model study was conducted and validated with the work of Binder (2013). 85 cases with different orifice length to diameter ratios, radius to diameter ratios and pressure ratios were run and 34 cases with inlet cross-flow were calculated with varying length to diameter ratio, radius to diameter ratio and cross-flow ratio. A comprehensive study of the results was done with comparison with previous articles and correlations, and the obtained result showed good agreement regarding the flow behaviour and discharge coefficient. Furthermore, the CFD approach offered more insight into the flow behaviour since the field variables can be visualized more easily as compared with experimental studies. Finally, all the discharge coefficients were collected into three correlations, one for the cases without inlet cross-flow and two for the cases with inlet cross-flow. The correlations have excellent performance in their defined range but should not be used outside the range and a number of suggestions are presented for the use of the correlations outside the range.
Table of Contents

List of Figures ........................................................................................................... 5
List of Tables .............................................................................................................. 8
Nomenclature ............................................................................................................ 9

1 Introduction ............................................................................................................ 11

2 Earlier studies and current goals ............................................................................. 12

3 Literature review ................................................................................................... 13
  3.1 Characterizing the losses .................................................................................... 13
  3.2 Parameters influencing the losses in orifices ..................................................... 14
    3.2.1 General description of the flow field ............................................................ 14
    3.2.2 The geometry of the orifice ........................................................................ 14
    3.2.3 Flow related parameters ............................................................................. 16
  3.3 Choking ................................................................................................................ 20
  3.4 Hysteresis ............................................................................................................ 21
  3.5 Correlations for the discharge coefficient .......................................................... 21

4 Theoretical background ........................................................................................... 23
  4.1 Governing equations in fluid mechanics ............................................................ 23
    4.1.1 Conservation of mass .................................................................................. 23
    4.1.2 Conservation of momentum ....................................................................... 23
    4.1.3 Conservation of energy .............................................................................. 24
    4.1.4 Equation of state ........................................................................................ 24
  4.2 Turbulence ........................................................................................................... 25
    4.2.1 The nature of turbulence ............................................................................ 25
  4.3 Turbulence modelling .......................................................................................... 25
    4.3.1 Reynolds averaged equations ...................................................................... 25
    4.3.2 The turbulent boundary layer ..................................................................... 27
    4.3.3 Boussinesq assumption .............................................................................. 28
    4.3.4 Two equation models ................................................................................ 29
    4.3.5 Reynolds Stress Models (RSM) and Large Eddy Simulations (LES) .......... 30
  4.4 Numerical Schemes ............................................................................................. 31

5 Methodology ........................................................................................................... 33
  5.1 Parameters of study ............................................................................................ 33
  5.2 Geometry ............................................................................................................. 34
  5.3 Computational mesh .......................................................................................... 36
    5.3.1 Mesh type I, sharp-edged orifices ................................................................. 36
    5.3.2 Mesh type II, radiused orifices ................................................................... 37
    5.3.3 Mesh type III, cross-flow ............................................................................ 38
  5.4 General Parameters ............................................................................................ 38
  5.5 Boundary conditions .......................................................................................... 39
    5.5.1 No cross-flow .............................................................................................. 39
    5.5.2 Cross-flow ................................................................................................... 41
  5.6 Mesh verification ................................................................................................ 42
    5.6.1 No cross-flow .............................................................................................. 42
    5.6.2 Cross-flow ................................................................................................... 51
  5.7 Solver validation ................................................................................................ 51
    5.7.1 Turbulence models ....................................................................................... 52
5.7.2 Model validation ................................................................. 54

6 Results – Study I, no cross-flow .............................................. 58
6.1 General overview ............................................................... 58
6.2 Detailed analysis ................................................................. 60
  6.2.1 L/D and $p_1/p_2$ .......................................................... 60
  6.2.2 L/D and r/D ................................................................. 61
  6.2.3 r/D and $p_1/p_2$ .......................................................... 63
  6.2.4 L/D, r/D and $p_1/p_2$ .................................................... 64
  6.2.5 Additional findings ....................................................... 65
6.3 Comparison with chamfered orifices .................................... 67
6.4 Comparison with correlations in literature ............................ 68

7 Results – Study II, cross-flow .................................................. 71
7.1 General overview ............................................................... 72
7.2 Detailed analysis ................................................................. 73
  7.2.1 L/D and $U_{ca}$ ........................................................... 73
  7.2.2 L/D and r/D ................................................................. 74
  7.2.3 r/D and $U_{ca}$ ............................................................. 75
  7.2.4 L/D, r/D and $U_{ca}$ ....................................................... 77
7.3 Comparison with data from inlet cross-flow and rotating orifices... 78
7.4 Comparison with correlations in literature ............................ 80

8 Correlation ............................................................................. 82
8.1 Correlation- Study I ............................................................. 82
8.2 Correlations-Study II .......................................................... 84
8.3 Concluding remarks ........................................................... 86

9 Conclusion ............................................................................. 88

Bibliography ............................................................................ 89
Appendix A .............................................................................. 91
Appendix B .............................................................................. 92
Appendix C .............................................................................. 95
List of Figures

Figure 3.1: Overview of orifice geometry. Adapted from Idris & Pullen (2005) .......................... 14
Figure 3.2: Schematic of flow through an orifice with inlet cross-flow. Adapted from Idris & Pullen (2005) .................................................................................................................. 17
Figure 3.3: Vector field of a rotating orifice seen from the orifice's frame of reference (Dittman et al., 2003) .................................................................................................................. 18
Figure 5.1: Schematic diagram of test rig (Binder et al., 2014) ......................................................... 34
Figure 5.2: Geometry of computational domain L/D = 4.0, r/D = 0.08. (a) Study I. (b) Study II. ................................................................................................................................. 35
Figure 5.3: L/D = 4.0, r/D = 0. (a) Structured hexahedral mesh. (b) Unstructured tetrahedral mesh, Hemisphere I. .......................................................................................................................... 37
Figure 5.4: L/D = 4.0, r/D = 0. (a) Unstructured tetrahedral mesh, Hemisphere II. (b) Final mesh. .................................................................................................................................................. 37
Figure 5.5: L/D = 4.0, r/D = 0.08. (a) Structured hexahedral mesh. (b) Final mesh. ................. 37
Figure 5.6: L/D = 4.0, r/D = 0, cross-flow. (a) Structured hexahedral mesh adapted for cross-flow. (b) Final mesh. ........................................................................................................ 38
Figure 5.7: L/D = 4.0, r/D = 0.08, cross-flow. Unstructured tetrahedral mesh, Cross-flow cube. .............................................................................................................................. 38
Figure 5.8: Boundary conditions. L/D = 4.0, r/D = 0.08 ............................................................... 40
Figure 5.9: Boundary conditions, cross-flow. ............................................................................... 41
Figure 5.10: Separation region illustrated by contour plots of negative axial velocity. (a) Mesh 1. (b) Mesh 4. ........................................................................................................................................ 44
Figure 5.11: Lines across the orifice ............................................................................................. 44
Figure 5.12: Axial velocity profile across the orifice at x/L = 0.095 ............................................. 45
Figure 5.13: Turbulent kinetic energy profile across the orifice at x/L = 0.095 ......................... 45
Figure 5.14: Axial velocity profile across the orifice at x/L = 0.952 ............................................ 46
Figure 5.15: Turbulent kinetic energy profile across the orifice at x/L = 0.952 ......................... 46
Figure 5.16: Axial velocity profile across the orifice at x/L = 0.095, Mesh 1. ......................... 47
Figure 5.17: Contour plot of Mach number for Mesh 4, pressure ratio 1.5 ................................. 48
Figure 5.18: Separation region illustrated by negative axial velocity contours. L/D = 0.4, r/D = 0.04, Mesh 5 .................................................................................................................. 49
Figure 5.19: Axial velocity profile across the orifice at x/L = 0.4 ............................................... 50
Figure 5.20: Turbulent kinetic energy profile across the orifice at x/L = 0.4 ............................. 50
Figure 5.21: Axial velocity profiles across the orifice at x/L = 0.4, Mesh 5. .............................. 51
Figure 5.22: Validation case 1. L/D = 0.4, r/D = 0. ..................................................................... 55
Figure 5.23: Recirculation region illustrated by the contour plot of the negative axial velocities. (a) L/D = 0.4, r/D = 0, p_1/p_2= 1.3. (b) L/D = 0.4, r/D = 0, p_1/p_2= 1.65 ............................. 55
Figure 5.24: Validation case 2. L/D = 1.5, r/D = 0. ..................................................................... 56
Figure 5.25: Validation case 3, L/D = 2.0, r/D = 0, p_1/p_2...................................................... 57
Figure 6.1: (a) Discharge coefficient C_D as a function of pressure ratio p_1/p_2, L/D = 0.4. (b) Discharge coefficient C_D as a function of pressure ratio p_1/p_2, L/D = 2.0 ................................................. 59
Figure 6.2: (a) Discharge coefficient C_D as a function of pressure ratio p_1/p_2, L/D = 4.0. (b) Discharge coefficient C_D as a function of length to diameter ratio L/D, r/D = 0.04 ................................. 59
Figure 6.3: (a) Discharge coefficient C_D versus the pressure ratio p_1/p_2, r/D = 0.04. (b) Discharge coefficient C_D versus the length to diameter ratio L/D, r/D = 0.04. .............................. 61
Figure 6.4: Recirculation region illustrated by contour plots of the negative axial velocity. (a) L/D = 0.4, r/D = 0.04, p_1/p_2 = 1.4. (b) L/D = 2.0, r/D = 0.04, p_1/p_2 = 1.4 ............................ 61
Figure 6.5 (a) Discharge coefficient $C_D$ versus the radius to diameter ratio $r/D$, $p_1/p_2 = 1.1$. (b) Discharge coefficient $C_D$ versus the length to diameter ratio $L/D$, $p_1/p_2 = 1.1$. .......................... 62
Figure 6.6: Recirculation region illustrated by contour plots of the negative axial velocities. (a) $L/D = 0.4$, $r/D = 0.16$, $p_1/p_2 = 1.1$. (b) $L/D = 0.4$, $r/D = 0.20$, $p_1/p_2 = 1.1$.......................... 62
Figure 6.7 (a) Discharge coefficient $C_D$ versus the pressure ratio $p_1/p_2$, $L/D = 2.0$. (b) Discharge coefficient $C_D$ versus the radius over diameter ratio $r/D$, $L/D = 2.0$................. 63
Figure 6.8: Recirculation region illustrated by a contour plot of negative axial velocities. $L/D = 2.0$, $r/D = 0.16$, $p_1/p_2 = 1.4$. .................................................. 64
Figure 6.9: Discharge coefficient $C_D$ versus radius to diameter ratio $r/D$, $L/D = 0.4$................. 64
Figure 6.10: Recirculation region illustrated by contour plots of negative axial velocities. (a) $L/D = 0.4$, $p_1/p_2 = 1.4$, $r/D = 0.16$. (b) $L/D = 0.4$, $p_1/p_2 = 1.4$, $r/D = 0.20$......................... 65
Figure 6.11: Discharge coefficient $C_D$ versus pressure ratio $p_1/p_2$, $r/D = 0.16$. .......................... 65
Figure 6.12: Comparison of unsteadiness of the recirculation region, $L/D = 0.4$, $r/D = 0.16$, $p_1/p_2 = 1.05$. (a) Instantaneous axial velocity at 0.01 s. (b) Time-averaged axial velocity...... 66
Figure 6.13: Comparison between chamfered and radiused orifices, $L/D = 0.4$......................... 67
Figure 6.14: Comparison between chamfered and radiused orifices, $L/D = 2.0$-2.14. ............... 68
Figure 6.15: Comparison of data with correlations. (a) $L/D = 0.4$, $r/D = 0.04$. (b) $L/D = 2.0$, $r/D = 0.04$.................................................. 69
Figure 6.16: Comparison of data with correlations, $L/D = 4.0$, $r/D = 0.04$.......................... 70
Figure 7.1: Static discharge coefficient $C_{D,s}$. (a) $L/D = 2.0$. (b) $L/D = 4.0$. ......................... 72
Figure 7.2: Total discharge coefficient $C_{D,t}$. (a) $L/D = 2.0$. (b) $L/D = 4.0$. ......................... 73
Figure 7.3 (a) Static discharge coefficient $C_{D,s}$ versus the cross-flow ratio $U/c_{ax}$, $r/D = 0.08$. (b) Static discharge coefficient $C_{D,s}$ versus the length to diameter ratio $L/D$, $r/D = 0.08$............ 74
Figure 7.4: Recirculation region illustrated by contour plots of negative axial velocities. (a) $L/D = 2.0$, $r/D = 0.08$, $U/c_{ax} = 0.689$. (b) $L/D = 4.0$, $r/D = 0.08$, $U/c_{ax} = 0.689$....................... 74
Figure 7.5: (a) Static discharge coefficient $C_{D,s}$ versus the radius to diameter ratio $r/D$, $U/c_{ax} = 0.259$. Right: Static discharge coefficient $C_{D,s}$ versus the length to diameter ratio $L/D$, $U/c_{ax} = 0.259$.......................... 75
Figure 7.6: (a) Static discharge coefficient $C_{D,s}$ versus the cross flow ratio $U/c_{ax}$, $L/D = 4.0$. (b) Static discharge coefficient $C_{D,s}$ versus the radius to diameter ratio $r/D$, $L/D = 4.0$........... 76
Figure 7.7: Stagnation region, $L/D = 4.0$, $r/D = 0.30$, $U/c_{ax} = 1.12$. (a) Velocity contour plot. (b) Static pressure contour plot. .................................................. 76
Figure 7.8: Pressure ratio $p_{orifice}/p_2$ at $x/L = 0.75$ and $x/L = 1.0$ for $L/D = 4.0$, $r/D = 0.30$ .. 77
Figure 7.9: Static discharge coefficient $C_{D,s}$ versus the length to diameter ratio $L/D$, $U/c_{ax} = 1.12$.......................... 77
Figure 7.10: Comparison with Rohde, et al. (1969). (a) $L/D = 1.06-2.0$, $r/D = 0.1951-0.4878$. (b) $L/D = 4.0$, $r/D = 0$.................................................. 78
Figure 7.11: Comparison with Jakoby, et al. (1997). (a) $L/D = 2.0-2.66$, $r/D = 0.08-0.1$. (b): $L/D = 2.0-2.66$, $r/D = 0.30$.................................................. 79
Figure 7.12: Comparison with Dittman et al. (2003), $L/D = 1.25-2.0$, $r/D = 0-0.20.$ ............... 80
Figure 7.13: Comparison with Idris and Pullen correlation. (a) $L/D = 2.0$, $r/D = 0-0.30$. (b) $L/D = 4.0$, $r/D = 0-0.30$.................................................. 81
Figure 8.1: Discharge coefficients given by the correlation versus the discharge coefficients from the current study. .................................................. 83
Figure 8.2: Comparison of intermediate variable values with CFD data. (a): $L/D = 2.0$, $r/D = 0-0.08$. (b): $L/D = 2.0-4.0$, $r/D = 0.30$.................................................. 84
Figure 8.3: Static discharge coefficients $C_{D,s}$ given by the correlation versus the static discharge coefficients $C_{D,s}$ from the current study. .................................................. 85
Figure 8.4: Total discharge coefficients $C_{D,t}$ given by the correlation versus the total discharge coefficients $C_{D,t}$ from the current study. .................................................. 86
Figure 8.5: Comparison of intermediate variable values with CFD data. L/D = 2.0, r/D = 0.08-0.30 (a): Static discharge coefficient $C_{D,s}$. (b): Total discharge coefficient $C_{D,t}$.

Figure B. 1: Axial velocity profile across the orifice at $x/L = 0.476$. L/D = 1.5, r/D = 0.

Figure B. 2: Turbulent kinetic energy profile across the orifice at $x/L = 0.476$. L/D = 1.5, r/D = 0.

Figure B. 3: Axial velocity profile across the orifice at $x/L = 0.6$. L/D = 0.4, r/D = 0.04.

Figure B. 4: Turbulent kinetic energy profile across the orifice at $x/L = 0.6$. L/D = 0.4, r/D = 0.04.

Figure B. 5: Axial velocity profile across the orifice at $x/L = 0.8$. L/D = 0.4, r/D = 0.04.

Figure B. 6: Turbulent kinetic energy profile across the orifice at $x/L = 0.8$. L/D = 0.4, r/D = 0.04.
List of Tables

Table 5.1: Orifice geometry, Study I.................................................................33
Table 5.2: Orifice geometry, Study II ...........................................................34
Table 5.3: Dimensions of parts in the computational domain..........................35
Table 5.4: Solver settings ..............................................................................39
Table 5.5: Boundary conditions, no cross-flow..............................................40
Table 5.6: Boundary conditions, cross-flow ..................................................42
Table 5.7: Mesh verification. L/D = 1.5, r/D = 0, mesh parameters ..............43
Table 5.8: Average element volume comparison normalized with the element volume of Mesh 3 ..........................................................43
Table 5.9: Mesh verification. L/D = 1.5, r/D = 0, results.................................43
Table 5.10: Relative difference between Mesh 1 and Mesh 4 in axial velocity and TKE at x/L = 0.095 .................................................................47
Table 5.11: Mesh verification. L/D = 0.4, r/D = 0.04, mesh parameters ..........48
Table 5.12: Mesh verification. L/D = 0.4, r/D = 0.04, results..........................49
Table 5.13: Relative difference between Mesh 5 and Mesh 7 in axial velocity and TKE at x/L = 0.4.................................................................51
Table 5.14: Geometrical and physical parameters for turbulence model study ..........................52
Table 5.15: Overview of studied turbulence models ........................................52
Table 5.16: Relative difference in mass flow with Binder (2013) .....................53
Table 5.17: Summary of validation cases. .......................................................54
Table 6.1: Summary of flow cases in Study I ..................................................58
Table 6.2: Pressure drop across the orifice .....................................................60
Table 6.3: Flow cases with supersonic regions ..............................................66
Table 6.4: Maximum and mean relative difference of correlations .................70
Table 7.1: Summary of flow cases in Study II ...............................................71
Table 7.2: Max and mean relative difference of correlations .........................81
Table 8.1: Overview of correlations ..............................................................87
Table A. 1: Overview of the meshes used in chapters 6 and 7 .......................91

Table C. 1: Coefficients a_{ijk} ....................................................................95
Table C. 2: Coefficients b_{ijk} ....................................................................95
Table C. 3: Coefficients c_{ijk} ....................................................................96
## Nomenclature

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>$A_0$</td>
<td>Orifice area [$m^2$]</td>
</tr>
<tr>
<td>$A$</td>
<td>Area [$m^2$]</td>
</tr>
<tr>
<td>$a_{ijk}, b_{ijk}, c_{ijk}$</td>
<td>Correlation constants [-]</td>
</tr>
<tr>
<td>$B$</td>
<td>Constant [-]</td>
</tr>
<tr>
<td>$c_{ax}$</td>
<td>Ideal axial velocity based on static pressure ratio [m/s]</td>
</tr>
<tr>
<td>$C_D$</td>
<td>Discharge coefficient [-]</td>
</tr>
<tr>
<td>$c_p$</td>
<td>Heat capacity at constant pressure [$J/K$]</td>
</tr>
<tr>
<td>$c_v$</td>
<td>Heat capacity at constant volume [$J/K$]</td>
</tr>
<tr>
<td>$C_{\mu}$</td>
<td>Model constant [-]</td>
</tr>
<tr>
<td>$C_{\varepsilon 1}$</td>
<td>Model constant [-]</td>
</tr>
<tr>
<td>$C_{\varepsilon 2}$</td>
<td>Model constant [-]</td>
</tr>
<tr>
<td>CFD</td>
<td>Computational Fluid Dynamics</td>
</tr>
<tr>
<td>$D$</td>
<td>Orifice diameter [$m$]</td>
</tr>
<tr>
<td>$E_t$</td>
<td>Total energy per unit volume [$J/m^3$]</td>
</tr>
<tr>
<td>$e$</td>
<td>Internal energy per unit volume [$J/m^3$]</td>
</tr>
<tr>
<td>$f$</td>
<td>Darcy friction factor [-]</td>
</tr>
<tr>
<td>$f$</td>
<td>Body force per unit mass vector [N/kg]</td>
</tr>
<tr>
<td>$g$</td>
<td>Variable</td>
</tr>
<tr>
<td>$(g)$</td>
<td>Gauge</td>
</tr>
<tr>
<td>$H$</td>
<td>Total enthalpy per unit volume [$J/m^3$]</td>
</tr>
<tr>
<td>$k$</td>
<td>Turbulent kinetic energy [$m^2/s^2$]</td>
</tr>
<tr>
<td>$L$</td>
<td>Orifice length [$m$]</td>
</tr>
<tr>
<td>$M$</td>
<td>Mach number [-]</td>
</tr>
<tr>
<td>$\dot{m}$</td>
<td>Mass flow [kg/s]</td>
</tr>
<tr>
<td>$p$</td>
<td>Pressure [Pa]</td>
</tr>
<tr>
<td>$\Delta p$</td>
<td>Pressure loss [Pa]</td>
</tr>
<tr>
<td>PLC</td>
<td>Pressure loss coefficient [-]</td>
</tr>
<tr>
<td>$Q$</td>
<td>External heat addition per unit volume [$J/m^3$]</td>
</tr>
<tr>
<td>$q$</td>
<td>Heat flux vector [W/m$^2$]</td>
</tr>
<tr>
<td>$r$</td>
<td>Radius of inlet radiusing [m]</td>
</tr>
<tr>
<td>$R$</td>
<td>Specific gas constant [J/kg/K]</td>
</tr>
<tr>
<td>$Re$</td>
<td>Reynolds number [-]</td>
</tr>
<tr>
<td>RMS</td>
<td>Root mean square</td>
</tr>
<tr>
<td>$T$</td>
<td>Temperature [K]</td>
</tr>
<tr>
<td>$\Delta t$</td>
<td>Time scale [s]</td>
</tr>
<tr>
<td>TKE</td>
<td>Turbulent kinetic energy</td>
</tr>
<tr>
<td>$Tu$</td>
<td>Turbulence intensity [%]</td>
</tr>
<tr>
<td>$U$</td>
<td>Cross-flow velocity [m/s]</td>
</tr>
<tr>
<td>$u$</td>
<td>Velocity vector [m/s]</td>
</tr>
<tr>
<td>$u^+$</td>
<td>Dimensionless velocity [-]</td>
</tr>
<tr>
<td>$u_f$</td>
<td>Friction velocity [m/s]</td>
</tr>
<tr>
<td>$V$</td>
<td>Velocity vector magnitude [m/s]</td>
</tr>
<tr>
<td>$w$</td>
<td>Chamfer depth [m]</td>
</tr>
<tr>
<td>$w_{ax}$</td>
<td>Ideal axial velocity based on static pressure ratio [m/s]</td>
</tr>
</tbody>
</table>
\[ x \quad \text{Orifice axial distance [m]} \]
\[ y \quad (\text{Orifice) wall distance [m]} \]
\[ y^+ \quad \text{Non dimensional wall distance [-]} \]
\[ z \quad \text{Orifice wall distance [m]} \]

**Greek symbols**

\[ \alpha \quad \text{Model constant [-]} \]
\[ \beta, \beta' \quad \text{Model constant [-]} \]
\[ \gamma \quad \text{Heat capacity ratio [-]} \]
\[ \Gamma_T \quad \text{Turbulent diffusivity [Pa s]} \]
\[ \delta \quad \text{Boundary layer thickness [m]} \]
\[ \delta_{ij} \quad \text{Kronecker delta, } \delta_{ij} = 1 \text{ if } i=j, \text{ else } \delta_{ij} = 0 \]
\[ \delta_v \quad \text{Viscous length-scale [m]} \]
\[ \epsilon \quad \text{Turbulent eddy dissipation [m}^2/\text{s}^3] \]
\[ \theta_v \quad \text{Velocity head ratio [-]} \]
\[ \kappa \quad \text{Von Kármán constant [-]} \]
\[ \lambda \quad \text{Thermal conductivity [W/m/K]} \]
\[ \mu \quad \text{Dynamic viscosity [Pa s]} \]
\[ \mu_T \quad \text{Turbulent viscosity [Pa s]} \]
\[ \nu \quad \text{Kinematic viscosity [m}^2/\text{s} \]
\[ \Pi_{ij} \quad \text{Stress tensor} \]
\[ \rho \quad \text{Density [kg/m}^3 \]
\[ \sigma_{ij}, \sigma_v, \sigma_{\omega} \quad \text{Model constant [-]} \]
\[ \tau \quad \text{Viscous stress tensor} \]
\[ \tau_w \quad \text{Wall shear stress [Pa]} \]
\[ \phi \quad \text{Scalar variable} \]
\[ \omega \quad \text{Large scale eddy frequency [s}^{-1} \]

**Superscripts**

\[ ^{\circ} \quad \text{Degree} \]
\[ _{-} \quad \text{Mean} \]
\[ _{'} \quad \text{Fluctuating} \]
\[ _{*} \quad \text{Modified} \]

**Subscripts**

\[ 1 \quad \text{Upstream of orifice} \]
\[ 2 \quad \text{Downstream of orifice} \]
\[ \text{abs} \quad \text{Absolute} \]
\[ \text{avg} \quad \text{Average} \]
\[ \text{crit} \quad \text{Critical} \]
\[ \text{eff} \quad \text{Effective} \]
\[ \text{ideal} \quad \text{Isentropic process} \]
\[ \text{i,j,k} \quad \text{Index notation} \]
\[ \text{major} \quad \text{Major} \]
\[ \text{minor} \quad \text{Minor} \]
\[ \text{r} \quad \text{Radiused} \]
\[ \text{rel} \quad \text{Relative} \]
\[ \text{rot} \quad \text{Rotation} \]
\[ s \quad \text{Static} \]
\[ t \quad \text{Total} \]
1 Introduction

The demand for higher thermal efficiency in gas turbines results in higher operating temperatures; consequently the materials are subjected to an increased thermal load which can exceed the critical values and affect the component lifetime negatively. As a result, heat must be transferred away through an internal flow network commonly named the secondary air system. Cooling air is bled from the compressor and fed into the secondary air system. Flow restrictors, such as circular orifices are used to control the flow to ensure a correct flow distribution in the secondary air system. The challenge is to minimize the bleed air from the compressor since it does not contribute to the turbine output while it still requires compressional work. This requires a thorough understanding of the flow through the passages and orifices in the secondary air system to correctly distribute the cooling air since an imbalanced flow distribution might lead to premature component failure.

Orifices are commonly employed in the secondary air system and the objective of this work is to investigate the flow behaviour, more specifically the mass flow and pressure losses, of orifices through the use of Computational Fluid Dynamics (CFD). The work is a continuation of the work by Binder (2013) and (Binder et al., 2014) and was conducted for Siemens Industrial Turbomachinery, Finspång Sweden.
2 Earlier studies and current goals

Binder (2013) conducted an experimental study on a stationary engine-like test rig at Siemens Industrial Turbomachinery, Finspång Sweden, to investigate the effects of how different parameters, physical and geometrical, influence the flow through the orifices found in secondary air systems. It was deduced after gathering all the relevant parameters and using Buckingham $\pi$ analysis that three dimensionless groups of interest were feasible for the experimental study, the length to diameter ratio of the orifice $L/D$, the chamfering depth to diameter ratio of the orifice $w/D$ and the pressure ratio $p_1/p_2$. The study produced a correlation for the discharge coefficient based on the dimensionless groups. In real life applications, radiused orifices are also found in turbomachinery, thus the goal with the current thesis is to supplement the previous work with data from CFD calculations regarding radiused orifices. This introduces a new dimensionless parameter radius to diameter ratio $r/D$. Additionally, the effect of inlet cross-flow is investigated for sharp-edged and radiused orifices. Consequently a new dimensionless parameter cross-flow ratio $U/c_{ax}$ is introduced.

The experimental work was presented in an article (Binder et al., 2014) and references will be made to the article whenever possible due to the accessibility of the article compared to the diploma work (Binder, 2013).
3 Literature review

3.1 Characterizing the losses

When designing a pipe for internal flow, it is crucial to know the pressure losses arising in the flow. The pressure losses can be divided into two categories, major and minor losses. Major losses are the losses that occur due to the viscosity in the fluid and the process is irreversible. The expression for the pressure loss is given in Equation (3.1) without derivation (Çengel & Cimbala, 2006).

\[ \Delta p_{\text{major}} = f \frac{L \rho V_{\text{avg}}^2}{D^2/2} \]  

The Darcy friction factor \( f \) can be obtained for example in a Moody chart. This pressure loss can also be expressed as a head loss which is the extra static head \( h_L \) needed to overcome the pressure losses. This is illustrated in Equation (3.2).

\[ h_{L,\text{major}} = \frac{\Delta p_{\text{major}}}{\rho g} = f \frac{L V_{\text{avg}}^2}{D^2/2} \]  

Minor losses are the pressure losses that arise due to sudden changes in geometry such as inlets, bends and exits for example. The flow separates and additional turbulent mixing arises which induces an irreversible pressure loss. An example is when a flow enters a sudden contraction; the flow will accelerate and separate at the smallest cross-sectional area, also known as the vena contracta. According to Bernoulli’s principle, an acceleration of flow velocity will yield a decrease in static pressure. When the flow decelerates after the vena contracta, the static pressure increase according to the same principle but some of the pressure energy has been lost due to turbulent mixing. This pressure loss can be expressed as a pressure loss coefficient \( PLC \) which is defined as:

\[ \Delta p_{\text{minor}} = PLC \frac{\rho V^2}{2} \]  

It can be seen as the quotient of the difference in pressure energy per unit volume divided by the kinetic energy per unit volume.

An alternative way to quantify the pressure losses, both major and minor losses, is to define a compressible discharge coefficient \( C_D \) which is the ratio of the mass flow divided by the ideal mass flow that is achieved in an isentropic process.

\[ C_D = \frac{m}{m_{\text{ideal}}} \]  

Applying isentropic relations to find the ideal mass flow, the discharge coefficient \( C_D \) can be expressed as:

\[ C_D = \frac{m}{A_p \frac{p_{1t}^{2\gamma}}{\sqrt{RT_{1t}}} \left[ \frac{2\gamma}{\gamma - 1} \left( \frac{p_{25}^2}{p_{1t}} - \frac{p_{25}}{p_{1t}} \right)^{1+\gamma} \right]} \]  

The advantage in utilizing the discharge coefficient is that only the mass flow, pressure and temperature need to be measured whereas the pressure loss coefficient \( PLC \) requires the
measurement of velocity and density which is more complicated since the quantities are not constant for compressible flows and the point of evaluation of these quantities is not specified (Binder, 2013). Furthermore the compressible discharge coefficient embodies both major and minor losses in one expression. For this work, the compressible discharge coefficient $C_D$ will be the quantity used for evaluating the losses. The compressible discharge coefficient will henceforth be referred to as the discharge coefficient and all exceptions will be explicitly stated.

### 3.2 Parameters influencing the losses in orifices

There are several parameters, both geometrical and physical, which affect the pressure losses and consequently the discharge coefficient. A review of the available literature is given in regard to the parameters that influence the discharge coefficient.

#### 3.2.1 General description of the flow field

Flow entering orifices will separate at the inlet due to the sharp turn experienced by the flow. The amount of separation depends both on geometrical parameters but also on physical parameters, this also applies to whether the flow will reattach to the orifice wall or stay separated to the exit. If full reattachment occurs, a recirculation zone is created where flow is entrapped (Ward-Smith, 1979). In case where no reattachment occurs, the recirculation zone is not enclosed but open to the outlet of the orifice and flow is drawn into the recirculation region (Lichtarowicz et al., 1965).

#### 3.2.2 The geometry of the orifice

Orifices come in all forms and shapes; they can be round or rectangular, have a straight, chamfered or radiused inlet/outlet, and have a varying diameter to mention some properties. In this work, round orifices with constant diameter with and without radiusing are studied. To be able to generalize the study of orifices, the various parameters can be described by dimensionless groups. The length to diameter ratio $L/D$ describes the ratio between the axial length $L$ and the orifice diameter $D$. The chamfering depth to diameter ratio $w/D$ describes the ratio between the axial depth of the chamfering and the orifice diameter. The chamfering can have different angles; Binder et al. (2014), which this study connects to, used a chamfering angle of 45° but other chamfering angles have been studied (Hay & Spencer, 1992). The radius to diameter ratio $r/D$ describes the rounding radius to diameter ratio. A schematic of these geometrical parameters can be seen in Figure 3.1.

![Figure 3.1: Overview of orifice geometry. Adapted from Idris & Pullen (2005).](image)
3.2.2.1 Length to diameter ratio L/D
Lichtarowicz et al. (1965) concluded that the incompressible discharge coefficient rises sharply for $0.5 \leq L/D \leq 1.0$ with a further increase to $L/D = 2.0$ and then decreases almost linearly as $L/D$ becomes larger. They observed that full reattachment occurs for $L/D = 4.0$. Deckker & Chang (1965) found, through the use of a static pressure traverse, evidence that for $L/D = [1.0,2.0]$, pressure recovery existed for compressible flow. Hüning (2008) states that for $L/D > 2$, flow reattachment occurs which enables pressure recovery and increases $C_D$ as compared with short orifices without flow reattachment.

3.2.2.2 Chamfer depth to diameter ratio w/D
Orifices can be manufactured with a chamfer usually of 45° or 30°, but other angles do exist. The chamfer helps the flow to enter the orifice axially thereby reducing the separation and consequently the inlet losses. Hay & Spencer (1992) found experimentally that the discharge coefficient increased up to 22%, for a pressure ratio of 1.6 and for orifices with $0.25 \leq L/D \leq 2.0$ with a chamfering angle of 45°. At least 95% of the increase (22%) occurred at $w/D = 0.04$. The size of the vena contracta appears constant for $w/D \geq 0.08$ and therefore higher chamfering is not more beneficial for the discharge coefficient. Also, a chamfering angle of 30° showed to be more beneficial for the discharge coefficient in comparison with the 45° chamfer angle. It exhibited largely the same behaviour as its counterpart but with a systematically larger discharge coefficient when compared for different values of $L/D$. Binder et al. (2014) found a non-linear increase of the discharge coefficient when increasing $w/D$; this was more pronounced for $L/D < 2.14$. Binder et al. (2014), Dittman et al. (2003) and Hay & Spencer (1992) observed that for $L/D \leq 0.5$, excessive chamfering has negative effect on the discharge coefficient. Hay & Spencer (1992) proposed that this is due to marginal reattachment. The effective length of the orifice is reduced when the chamfer depth is increased, thereby influencing the possibility of reattachment and consequently pressure recovery which is associated with an increase in discharge coefficient. Binder et al. (2014) therefore proposed that the effective length of the orifice should be considered, defined as:

$$L_{eff} = \frac{L - w}{D} \quad (3.6)$$

Applying the definition, Binder et al. (2014) recommend from their measurements a $L_{eff} > 0.35$ to avoid the decrease in discharge coefficient. They apply the same definition on the data by Dittman et al. (2003) and obtains a similar value of $L_{eff} > 0.3$. However, when applying this to the data of Hay & Spencer (1992), the value of the smallest $L_{eff}$ range from 0.05 to 0.8.

3.2.2.3 Radiusing to diameter ratio r/D
Radiusing has a similar effect on the discharge coefficient as chamfering, i.e. the discharge coefficient is increased when applying radiusing. Hay & Spencer (1992) observed experimentally that for $L/D = 0.25$, small chamfering with a chamfer angle of 45° is superior to small radiusing but noticed that large radiusing is more beneficial than large chamfering. However, by using an orifice with a 30° chamfer angle, the discharge coefficient was equal for both chamfering and radiusing for high values of $w/D$ and $r/D$. They also noted that that the chamfered orifices have less dependence on the pressure ratio than the radiused orifices. Dittman et al. (2003) observed an increase of the discharge coefficient up to 21% for $r/D = 0.05$ and an increase up to 39% for $r/D = 0.20$ compared with a sharp-edged orifice with $r/D = 0$. 
3.2.2.4 **Surface roughness**

No study has been found where the author(s) study the effect of wall roughness. However, there are several studies where the effects of long orifices are shown. Lichtarowicz et al. (1965) present a linearly decreasing incompressible discharge coefficient for $2.0 \leq L/D \leq 10.0$. This is presumably due to frictional losses, i.e. major losses. For compressible flows, Fanno friction effects start to become prominent for $L/D > 2.0$ leading to a decreasing discharge coefficient according to Parker & Kercher (1991). However in a review by Hay & Lampard (1996), the effect of friction and consequently surface roughness, is stated to be negligible for $L/D < 8.0$.

3.2.3 **Flow related parameters**

3.2.3.1 **Pressure ratio $p_1/p_2$**

The pressure ratio, denoted as $p_1/p_2$, is the ratio of orifice upstream static (stagnation) pressure to the orifice downstream static pressure. Deckker & Chang (1965) found that the discharge coefficient increases when the pressure ratio increases and the dependence is strong for pressure ratios up to 2. This was found to be more pronounced for short orifices. The same behaviour was also observed by Brain & Reid (1975), Hay & Spencer (1992) and Binder et al. (2014). Ward-Smith (1979) explained this as an effect of the compressibility of flow for high pressure ratios. When the pressure ratio increases, the size of the recirculation zone is suppressed (i.e. the area of the vena contracta increases) and the amount of separated flow decreases which yields a higher discharge coefficient.

3.2.3.2 **Reynolds number Re**

The Reynolds number, denoted as $Re$, is an important quantity in fluid mechanics. It expresses the ratio of inertial forces to viscous forces and is defined as:

$$ Re = \frac{\rho V D}{\mu} \quad (3.7) $$

Lichtarowicz et al. (1965) gives a recommendation that the incompressible discharge coefficient is constant for $Re > 2 \cdot 10^4$ for $2 < L/D < 10$. Deckker & Chang (1965) found experimentally that the compressible discharge coefficient is unaffected by the Reynolds number if the latter being larger than $10^4$.

3.2.3.3 **Inlet cross flow**

Inlet cross flow introduces a velocity perpendicular to the orifice; a schematic can be seen in Figure 3.2.
A new parameter is introduced, namely the ratio of the cross flow velocity $U$ perpendicular to the orifice, to the ideal axial velocity in the orifice $w_{ax}$. This is based on the total pressure upstream and given as (Hüning, 2012):

$$\frac{U}{w_{ax}} = \sqrt{\frac{1 - \left(\frac{p_{1,s}}{p_{1,t}}\right)^{\frac{y-1}{y}}}{1 - \left(\frac{p_{2,s}}{p_{1,t}}\right)^{\frac{y-1}{y}}}} \quad (3.8)$$

Another way of defining the cross flow ratio is by using ideal axial velocity $c_{ax}$ which is not based on the total pressure but rather the total pressure in the orifice direction, which is here equivalent to the static pressure (McGreehan & Schotsch, 1988). Equation (3.8) then becomes:

$$\frac{U}{c_{ax}} = \sqrt{\frac{1 - \left(\frac{p_{1,s}}{p_{1,t}}\right)^{\frac{y-1}{y}}}{1 - \left(\frac{p_{2,s}}{p_{1,s}}\right)^{\frac{y-1}{y}}}} \quad (3.9)$$

It might be noted that the cross flow velocity $U$ is the ideal cross-flow velocity instead of the actual cross-flow velocity.

Two counteracting effects generally arise when cross-flow at the inlet is present (Hüning, 2008):

1. The flow separation region in the orifice on the upstream flow side is augmented which leads to a lower through flow due to the reduction of the effective flow area.
2. The occurrence of flow stagnation in the orifice inlet which enables a partial recovery of the dynamic pressure from the cross-flow. The actual pressure that drives the flow through the orifice has a value between the total pressure in the orifice’s direction and the total pressure where the cross-flow dynamic pressure is included, this leads to an augmentation in orifice through flow.

There are two definitions of the discharge coefficient when inlet cross flow is present. The discharge coefficient can be defined without the inclusion of the tangential velocity.
component in the total pressure and temperature in equation (3.5) but rather the total quantities along the axis of the orifice, in this case equal to the static pressure and temperature (McGreehan & Schotsch, 1988). The discharge coefficients based on the static values for cross flows will be denoted as $C_{D,s}$:

$$C_{D,s} = \frac{\dot{m}}{A_o \sqrt{RT_1} \sqrt{2\gamma - 1 \left[ \left( \frac{p_{2s}}{p_{1s}} \right)^{\frac{2}{\gamma}} - \left( \frac{p_{2s}}{p_{1s}} \right)^{\frac{1+\gamma}{\gamma}} \right]}} \quad (3.10)$$

However, in many publications with cross flows where the orifices are inclined, the total quantities are used (Rohde et al., 1969; Hüning, 2012). This definition of the discharge coefficient will be denoted as $C_{D,t}$:

$$C_{D,t} = \frac{\dot{m}}{A_o \sqrt{RT_1} \sqrt{2\gamma - 1 \left[ \left( \frac{p_{2t}}{p_{1t}} \right)^{\frac{2}{\gamma}} - \left( \frac{p_{2t}}{p_{1t}} \right)^{\frac{1+\gamma}{\gamma}} \right]}} \quad (3.11)$$

Hüning (2012) found the total discharge coefficient $C_{D,t}$ to decrease for sharp-edged orifices with increasing cross flow ratios. He also noted a slight raise of the discharge coefficient with increasing static pressure ratio and also an augmentation of the discharge coefficient with increasing $L/D$. The results obtained were similar to the results by Rohde et al. (1969) and Dittman et al. (2003).

### 3.2.3.4 Rotation

Rotation is not studied in the current work but due to the fact that it is fairly similar to inlet cross flow, rotation will be treated as well. Focus will be on rotation with an axis parallel to the orifice axis.

![Vector field of a rotating orifice seen from the orifice’s frame of reference](Dittman et al., 2003).

From the orifice’s point of view, inlet and exit cross-flow is seen due to the rotation, this is illustrated by Figure 3.3. Similarly to orifices with inlet cross flow, there exist two definitions of the discharge coefficient. In the absolute frame, the absolute discharge coefficient $C_{D,abs}$ does not account for the rotational effects from a rotating orifice and only depends on total
values upstream and static values downstream of the orifice. Its form is identical to Equation (3.5). To account for the rotational effects, a relative discharge coefficient \( C_{D,rel} \) can be used which is based on the total pressure in the rotating (relative) frame, i.e. the frame of the rotating orifice. The relative total temperature and relative total pressure are given as:

\[
T_{1t,rel} = T_{1t} \cdot (1 + \frac{U_{rot}^2}{2c_p T_{1t}}) \quad (3.12)
\]

\[
P_{1t,rel} = P_{1t} \cdot \left(1 + \frac{U_{rot}^2}{2c_p T_{1t}}\right)^{\frac{\gamma}{\gamma-1}} \quad (3.13)
\]

\( U_{rot} \) is the tangential velocity of the orifice, consequently the orifice should experience a cross-flow velocity \( U_{rot} \). The relative discharge coefficient \( C_{D,rel} \) can now be expressed as (Dittman et al., 2003):

\[
C_{D,rel} = \frac{\dot{m}}{A_o \sqrt{RT_{1t,rel}} \sqrt{\frac{2\gamma}{\gamma-1} \left[\frac{(p_{2s}/p_{1t,rel})^{\frac{2}{\gamma}}}{\left(\frac{p_{2s}}{p_{1t,rel}}\right)^{\frac{1}{\gamma}}} - 1\right]}} \quad (3.14)
\]

The cross-flow ratio found in Equation (3.8) for inlet cross-flow is written in the relative frame for a rotating orifice as:

\[
U_{rot} = \frac{U_{rot}}{w_{ax}} = \frac{U_{rot}}{\sqrt{\frac{2\gamma}{\gamma-1} RT_{1t,rel} \left[1 - \left(\frac{p_{2s}}{p_{1t,rel}}\right)^{\frac{\gamma-1}{\gamma}}\right]}} \quad (3.15)
\]

Analogue with the previous example, the cross-flow ratio found in Equation (3.9) is for rotating orifices written in the absolute frame as (Dittman et al., 2003):

\[
\frac{U_{rot}}{C_{ax}} = \frac{U_{rot}}{\sqrt{\frac{2\gamma}{\gamma-1} RT_{1t} \left[1 - \left(\frac{p_{2s}}{p_{1t}}\right)^{\frac{\gamma-1}{\gamma}}\right]}} \quad (3.16)
\]

Hüning (2008) discusses the additional effects from centrifugal forces that are present for a rotating inlet flow or a rotating orifice but disregards this effect when using data from rotating orifices by Dittman et al. (2003) to present the effect of inlet cross-flow. Hüning (2012) experimentally studied inlet cross flow for a stationary orifice and stated that direct comparison can be made between the discharge coefficient based on total pressure ratio for inlet cross flow \( C_{D,t} \) in Equation (3.11) with the relative discharge coefficient \( C_{D,rel} \) given in Equation (3.14) when comparing his results with the data from Dittman et al. (2003) based on the discussion in his previous work (Hüning, 2008). This statement implicitly assumes that the definition of cross-flow velocity ratio as seen in Equations (3.8) and (3.15) can be compared directly as well. Applying the same reasoning as Hüning, the static discharge coefficient \( C_{D,s} \) defined in Equation (3.10) should be to some extent comparable to the discharge coefficient in the absolute frame \( C_{D,abs} \) as defined in equation (3.5) since neither of them consider the effect of cross-flow as seen by the orifice whether the orifice is stationary or rotating. Also, the cross-flow velocity ratio as defined in equation (3.16) should then be comparable to the cross-flow velocity ratio as defined in equation (3.9).
The effect of an axially rotating orifice has been studied by several authors. Jakoby et al. (1997) found that the absolute discharge coefficient $C_{D,\text{abs}}$ decreased for sharp orifices when increasing the rotational velocity since the flow becomes more separated in the orifice causing a decrease in the area of the vena contracta. However, an increase in the absolute discharge coefficient was observed for an orifice with $L/D = 2.66$, $r/D = 0.5$, with increasing rotational velocity but also an increase followed by a decrease for an orifice with $L/D = 2.66$, $r/D = 0.3$. This was explained as the effect of work transfer from the rotating orifice to the flow. If the orifice is long enough, the flow will exit parallel to the orifice axis. From the point of view of the rotating orifice, the flow has been deflected and work has then been done on the flow according to Euler’s pump equation. Thus two counteracting effects are present: The separation decreases the discharge coefficient whereas the rotation enhances the discharge coefficient but the latter is dependent on inlet geometry, being pronounced for large inlet radiusing. Dittman et al. (2003) studied the effect of chamfering and radiusing on the discharge coefficient for rotating orifices and observed similar trends to Jakoby et al. (1997). Hüning (2008) explained that the rotational work transfer as stated earlier can simply be seen as the “impact of flow stagnation in the relative system” thus, as understood by the author, enabling the comparison between orifices with an inlet cross-flow and rotating orifices.

### 3.3 Choking

Choked flow is the condition where changes in downstream pressure do not affect the flow field upstream. For one dimensional isentropic flow, the local axial velocity will reach sonic conditions at the smallest cross sectional area, being it a physical wall in a nozzle or a “wall” formed by a separation bubble. The “wall” formed by the separation bubble, commonly called vena contracta, acts as a solid wall as no information can propagate upstream. Gas dynamics gives the relation between total pressure and static pressure as:

$$\frac{p_t}{p_s} = \left(1 + \frac{\gamma - 1}{2} M^2\right)^{\frac{\gamma}{\gamma-1}} \tag{3.17}$$

For air, with the value of $\gamma = 1.4$, it can be deduced that for a Mach number larger than unity, the pressure ratio $p_t/p_s$ has to be larger than 1.8929, this is denoted as the critical pressure ratio. The flow is choked over this critical pressure ratio. The critical isentropic mass flow, is defined as:

$$m_{\text{ideal, crit}} = p_t A \sqrt{\frac{\gamma}{RT} \left(1 + \frac{\gamma - 1}{2}\right)^{\frac{\gamma+1}{2(\gamma-1)}}} \tag{3.18}$$

If the upstream total pressure is held constant, the mass flow is limited for all pressure ratios above critical pressure ratio if the walls are physical as the smallest cross-sectional area cannot be altered. For orifices where the separation of flow forms a vena contracta, the choking behaviour is different due to the fact that the size of the separation bubble can be altered by varying the downstream pressure. Also depending on the orifice geometry, choking can occur at the exit of the orifice. An excellent treatment of this subject can be found in Ward-Smith (1979).

The choked flow regime involves regions with transonic flow which presents an additional phenomenon, namely shocks. To capture shocks accurately with CFD, the computational mesh must be modelled with a sufficiently fine mesh at the shock region to accurately capture
the shock discontinuity. Due to time constraints, this was not investigated in this thesis; therefore this part will purposefully be kept brief.

The concept of the one-dimensional flow is an idealized treatment of flows in orifices and somewhat incomplete. Weir et al. (1956) found, using Schlieren visualization, oblique shock waves in two-dimensional orifices, and not the normal shock usually associated to the one-dimensional flow theory.

Deckker & Chang (1965) observed that for a very thin orifice, $L/D \sim 0$, the flow never chokes. Brain & Reid (1975) found that the flow does not choke for an orifice with $L/D = 0.14$ up to a pressure ratio of 5, furthermore they found choking to occur above the critical pressure ratio for $L/D = [0.28,0.31]$. Similarly Binder et al. (2014) found that choking for $L/D = 0.5$ occurs above the critical pressure ratio. On the other hand, Brain & Reid (1975) found that for $L/D > 1$, choking occurred below the critical pressure ratio. This was thought to be due to the pressure recovery in the orifice. The same behaviour for orifices with $L/D$ between 0.75 and 2.14 was observed by Binder et al. (2014).

### 3.4 Hysteresis

Several studies have found evidence of hysteresis in the discharge coefficient depending on whether the pressure ratio was increased or decreased. Deckker & Chang (1965) found evidence of hysteresis for $L/D = 0.5$ for sharp-edged orifices near the choking region; it was clearly manifested as two closed branches. The phenomenon was attributed to the attachment and detachment of the jet to and from the wall. Brain & Reid (1975) also observed hysteresis for sharp-edged orifices with $L/D = 0.5$ but not for orifices with $L/D < 0.5$. They also attributed this to jet reattachment. Binder et al. (2014) did not experience hysteresis for a sharp-edged orifice with $L/D = 0.5$ but experienced it when having a slight chamfering ($w/D = 0.00943$ and $w/D = 0.0169$). Furthermore, hysteresis was also observed for a sharp-edged orifice with $L/D = 0.75$ which has never been noticed before. Furthermore, they also observed hysteresis for low pressure ratios depending on whether the flow valve was opening or closing, i.e. whether the pressure ratio was increased or decreased. However, the findings are not consistent and they call for more investigations before hysteresis can be confirmed. The findings in this subchapter have all been found experimentally near the choking region and since this region is not treated in the current study, the effect of hysteresis will not be an issue. It is nonetheless good to have the knowledge of the hysteresis phenomenon associated with flow through orifices.

### 3.5 Correlations for the discharge coefficient

There are several correlations for the discharge coefficient. Lichtarowicz et al. (1965) provided a correlation for the incompressible discharge coefficient which regards length to diameter ratio and the Reynolds number. Parker & Kercher (1991) presented a correlation for the compressible discharge coefficient which included the effects of Reynolds number, radiusing and length to diameter ratio. Idris & Pullen (2005) provided an enhanced correlation including the effects of inlet chamfering, rotation and pumping effects due to rotation with an inclined orifice. Hünig (2008) provided a similar correlation to the one by Parker & Kercher (1991) where chamfering and inlet cross-flow was added. All previous correlations calculate the effect of the different parameters in an additive approach, where each parameter has a separate term. However, Binder et al. (2014) presented a correlation which includes the
pressure ratio, length to diameter ratio and inlet chamfer to diameter ratio where the effects of the parameters on each other are taken into account.
4 Theoretical background

In this chapter, the governing equations in fluid mechanics are presented; the first subchapter will treat the formulations of the governing equations. The second and third subchapter will present turbulence and subsequently its modelling. The final subchapter will briefly treat the numerical methods used for solving the governing equations in the current flow solver CFX.

4.1 Governing equations in fluid mechanics

In fluid mechanics, the governing equations are based on conservation laws, namely conservation of mass, momentum and energy. Additionally, an equation of state is necessary for joining the thermodynamic quantities pressure, density and temperature. For the interested reader, the details of the derivation for this subchapter can be found in Tannehill et al. (1997)

4.1.1 Conservation of mass

The conservation of mass law, also known as the continuity equation, can be seen in Equation (4.1):

\[ \frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{u}) = 0 \] (4.1)

The first term describes the density rate of change in a control volume and the second term describes the mass flux rate through the surface of the control volume. The material derivative is defined as:

\[ \frac{D}{Dt} = \frac{\partial}{\partial t} + \mathbf{u} \cdot \nabla \] (4.2)

Thus the conservation of mass law found in Equation (4.1) can be written as:

\[ \frac{D\rho}{Dt} + \rho (\nabla \cdot \mathbf{u}) = 0 \] (4.3)

For the incompressible flows, the first term vanishes since the density does not vary (assumption valid for \( M < 0.3 \)) and one obtains that the divergence of the velocity is equal to zero.

4.1.2 Conservation of momentum

The conservation of momentum can be derived from Newton’s second law and is given as:

\[ \frac{\partial (\rho \mathbf{u})}{\partial t} + \nabla \cdot (\rho \mathbf{uu}) = \rho \mathbf{f} + \nabla \cdot \mathbf{\Pi}_{ij} \] (4.4)

The rate of change of momentum in the control volume is described by the first term on the left hand side. The second term on the left hand side describes the momentum flux through the surface of the control volume. The body force per unit volume is described by the first term on the right hand side. The second term on the right hand side describes the surface force per unit volume applied on the fluid element and consists of shear and normal stresses in the
stress tensor $\Pi_{ij}$. By expanding the second term on the left hand side in Equation (4.4) and combining it with the continuity equation, the conservation of momentum can be expressed as:

$$\rho \frac{Du_i}{dt} = \rho f + \nabla \cdot \Pi_{ij} \tag{4.5}$$

Fluids that have a linear relation between stress and strain rate are called Newtonian fluids and the stress tensor $\Pi_{ij}$ can be written, as a function of pressure and velocity, in a tensor form as:

$$\Pi_{ij} = -p \delta_{ij} + \mu \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) - \frac{2}{3} \delta_{ij} \frac{\partial u_k}{\partial x_k} \tag{4.6}$$

The second term on the right hand side is usually called the viscous stress tensor $\tau_{ij}$. The final form of the momentum conservation equation, also known as the Navier-Stokes equation, can then be expressed as:

$$\rho \frac{Du_i}{dt} = \rho f - \nabla p + \frac{\partial}{\partial x_j} \left[ \mu \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) - \frac{2}{3} \delta_{ij} \mu \frac{\partial u_k}{\partial x_k} \right] \tag{4.7}$$

### 4.1.3 Conservation of energy

The conservation of energy can be found by applying the first law of thermodynamics on an infinitesimal fixed control volume. This yields the energy equation with $E_i$ being the total energy per unit volume:

$$\frac{\partial E_i}{\partial t} + \nabla \cdot (E_i \mathbf{u}) = \frac{\partial Q}{\partial t} - \nabla \cdot \mathbf{q} + \rho \mathbf{f} \cdot \mathbf{u} + \nabla \cdot (\Pi_{ij} \cdot \mathbf{u}) \tag{4.8}$$

The terms on the left hand side represent the rate of change of total energy in the control volume and the total energy flux through the boundaries respectively. The rate of heat from external sources is represented in the first term on the right hand side. The second term is the heat flux through the boundaries and can be expressed as:

$$\mathbf{q} = -\lambda \nabla T \tag{4.9}$$

which is known as Fourier's law for heat transfer where $\lambda$ is the thermal conductivity. The third and fourth term on the right hand side represents the work done on control volume by body and surface forces respectively.

### 4.1.4 Equation of state

The equation system formed by Equations (4.3), (4.7) and (4.8) contain seven scalar quantities, $\rho$, $p$, $e$, $T$, $u$, $v$, $w$, when regarding a Cartesian system and disregarding the body forces and external heat addition. However, the number of equations only amount to five scalar equations which presents a closure problem. For gases whose intermolecular forces are negligible, the perfect gas equation of state is valid:
\[ p = \rho RT \]  

where \( R \) is the specific gas constant. The assumption of a calorically perfect gas can be assumed for low temperature situations, this yields that the specific heat capacities at constant volume \( c_v \) is constant. The internal energy \( e \) can be defined as:

\[ e = c_v T \]

With these seven equations, the system can now be closed.

### 4.2 Turbulence

Turbulent flows are all around us in nature, observe for example the smoke from a cigarette or candle light, it ascends in a smooth manner initially until it suddenly behaves chaotically. In engineering applications, turbulent flows are prevalent, they occur for example in flows around vehicles, in pipes, in combustion to mention some areas.

#### 4.2.1 The nature of turbulence

Turbulent flows have a number of characteristics (Kundu & Cohen, 2008):

- They are chaotic.
- They have enhanced diffusivity of momentum and heat.
- They have vortices, also called eddies, with a large scale of sizes.
- Most of the energy is found in the largest eddies. The energy is transferred to the smaller eddies in a process known as the energy cascade. Non-linear viscous diffusion dissipates the energy in the smallest eddies. Thus a constant supply of energy is required for turbulent flows to balance the losses from viscous dissipation.

A famous example is the experiments carried out by Osbourne Reynolds. The flow of dye in a pipe showed that transition from laminar flow to turbulent flow occurs at a Reynolds number of around 2300-4000 (Pope, 2000).

### 4.3 Turbulence modelling

There are several ways to model turbulent flows. The current work uses a Reynolds averaged Navier Stokes (RANS) approach and thus, the focus of the following subchapters will be on this approach.

#### 4.3.1 Reynolds averaged equations

The continuity, momentum and energy equations are still valid for turbulent flows but since there are so many scales involved, solving the equations for all the scales presents quite a challenge for today’s computers. An alternative to solving for all scales is to solve for the mean flow characteristics averaged in time. This is known as Reynolds averaging which is obtained by the decomposition of the variables, e.g. \( g \), into a temporal mean component \( \bar{g} \) and
fluctuating component $g'$ followed by a time averaging of the governing equations. For compressible fluids, the density fluctuations have to be taken into account and the time averaging is weighed by the density, this is the case with the current flow solver CFX 14.5 (Solver Theory Guide, ANSYS Inc, 2012). For simplicity, the simpler Reynolds averaging will be presented here, as a consequence the small density fluctuations are not taken into account in the discussion. This subchapter is based on Tannehill et al. (1997) and the CFX Solver Theory Guide (ANSYS Inc, 2012). The definition of a time averaged mean quantity $\bar{g}$ is given as:

$$\bar{g} \equiv \frac{1}{\Delta t} \int_{t_0}^{t_0 + \Delta t} g \, dt$$  \hspace{1cm} (4.12)

The time scale $\Delta t$ has to be larger than the fluctuating motions but small compared to the unsteady flow field time scale. Time-averaging the fluctuating quantity $g'$ in a similar fashion as Equation (4.12) yields $\bar{g}' \equiv 0$. The variables can be written in a decomposed form.

$$u_j = \bar{u}_j + u'_j, \quad p = \bar{p} + p', \quad T = \bar{T} + T'$$ \hspace{1cm} (4.13)

Inserting the decompositions into the continuity equation as seen in Equation (4.1), and averaging the whole equation, the Reynolds-averaged continuity equation becomes:

$$\frac{\partial \bar{p}}{\partial t} + \frac{\partial}{\partial x_j} (\rho \bar{u}_j) = 0$$ \hspace{1cm} (4.14)

The momentum equation becomes:

$$\frac{\partial}{\partial t} (\rho \bar{u}_i) + \frac{\partial}{\partial x_j} (\rho \bar{u}_i \bar{u}_j) = - \frac{\partial \bar{p}}{\partial x_i} + \frac{\partial}{\partial x_j} (\bar{r}_{ij} - \rho \bar{u}_i' \bar{u}_j')$$ \hspace{1cm} (4.15)

The Reynolds-averaged momentum equation looks familiar save the addition of the term $-\rho \bar{u}_i \bar{u}_j$ also known as the Reynolds stress tensor. It is a symmetric tensor with nine components. When regarding the last term in equation (4.15), the first term describes the molecular diffusion of momentum while the second term can be taken as the macroscopic diffusion of momentum caused by turbulent fluctuations with the latter generally being larger than the former.

The energy equation has been expressed with the total energy $E$ previously. With the assumption of internal energy and kinetic energy being the only components of the total energy, the latter can be replaced with $\rho H + p$ where $H$ is the total enthalpy ($H = h + u_i u_i / 2$) and $h$ is the enthalpy ($h = c_p T$). Neglecting the rate of heat from external sources $\frac{\partial Q}{\partial t}$, Equation (4.8) can be rewritten as:

$$\frac{\partial}{\partial t} (\rho H) + \frac{\partial}{\partial x_j} (\rho u_j H + q_j - u_i \bar{r}_{ij}) = \frac{\partial p}{\partial t}$$ \hspace{1cm} (4.16)

The total enthalpy $H$ is also decomposed into a mean and fluctuating part and the final Reynolds averaged resulting energy equation becomes (CFX Solver Theory Guide, ANSYS Inc, 2012):
\[
\frac{\partial}{\partial t}(\rho \bar{H}) + \frac{\partial}{\partial x_j} \left( \rho \bar{u}_j \bar{H} + \rho u'_i h' - \lambda \frac{\partial T}{\partial x_j} \right) 
= \frac{\partial \bar{p}}{\partial t} + \frac{\partial}{\partial x_j} \left( \bar{u}_i (\bar{r}_{ij} - \rho \bar{u}_i \bar{u}_j') \right) 
\tag{4.17}
\]

It may be noted that \( \bar{H} = \bar{h} + \frac{1}{2} \bar{u}_i \bar{u}_i + \frac{1}{2} \bar{u}_i' \bar{u}_i'^2 \) where the last term is also known as the turbulent kinetic energy \( k \). It can be seen that there has been two terms added. In the second term on the right hand side, the Reynolds stress enters in the viscous work term and in the middle term in the second equation on the left hand side, the turbulent flux \( \rho u'_j h' \) has been added. These extra terms arising from Reynolds-averaging hands us a slight problem. There are more unknowns than equations and the equation system cannot be closed.

### 4.3.2 The turbulent boundary layer

Before tackling the problem with extra terms, it might be interesting to study the turbulent boundary layer. In wall bounded flows, a viscous fluid cannot slip against the wall, hence giving rise to a no-slip condition which imposes a zero flow velocity condition at the wall. This gives rise to a velocity gradient from the wall up to the free stream. One classical (laminar) example of this is the Blasius solution of the boundary layer on a flat plate where the velocity profile can be solved numerically (Kundu & Cohen, 2008).

The following part is based on the work of Pope (2000).

In turbulent flows, the boundary layer requires a separate treatment. Near the wall, the stresses that are given by the viscous stresses and Reynolds stresses, are dominated by the former one. As a result, the velocity profile is highly dependent on the Reynolds number. It might then be appropriate to define a viscous scale applicable in the near-wall region. Instead of using the index notation, Cartesian coordinates will be used for clarity. The friction velocity \( u_f \) is defined, with the help of wall shear stress \( \tau_w \) as:

\[
u_f \equiv \sqrt{\frac{\tau_w}{\rho}} \tag{4.18}\]

A viscous length scale \( \delta_v \) is defined as:

\[
\delta_v \equiv v \sqrt{\frac{\rho}{\tau_w}} = \frac{v}{u_f} \tag{4.19}
\]

The distance from the wall can be measured with the viscous length scale, commonly denoted as \( y^+ \) which is defined as:

\[
y^+ \equiv \frac{y}{\delta_v} = \frac{u_f y}{\nu} \tag{4.20}
\]

For \( y^+ < 50 \), usually called the viscous wall region, viscosity contributes significantly to the total shear stress. For \( y^+ > 50 \), the region is usually denoted as the outer layer and the effect of viscosity is negligible on the shear stress. For \( y^+ < 5 \), the region is called the viscous sublayer and the Reynolds stress has a negligible impact.

Prandtl postulated the existence of an inner layer, given a high Reynolds number, where the mean velocity is determined not by boundary layer thickness or free stream velocity but by
the viscous scales only. The inner layer spans from the wall to \( \frac{x}{\delta} = 0.1 \) where \( \delta \) is the boundary layer thickness. For \( y^+ < 5 \), i.e. the viscous sublayer, the dimensionless velocity \( u^+ = \frac{u}{u_e} \) can be taken as:

\[
    u^+ \equiv y^+
\]

This linear relation breaks down when \( y^+ > 12 \), where the deviation is larger than 25\%. For \( 30 < y^+ \), \( y/\delta < 0.3 \), a logarithmic velocity profile is valid:

\[
    u^+ \equiv \frac{1}{\kappa} \ln(y^+) + B \\
    \kappa = 0.41, B = 5.2
\]

The buffer layer is located between the viscous sublayer and the region where the logarithmic law holds and is the transition between the viscous and turbulent zones.

4.3.3 Boussinesq assumption

This subchapter is based on CFX Solver Theory Guide (ANSYS Inc, 2012). The Boussinesq assumption relates the Reynolds stress proportionally to the mean rate of strain:

\[
    -\rho \overline{u_i u_j} = \mu_t \left( \frac{\partial \overline{u_i}}{\partial x_j} + \frac{\partial \overline{u_j}}{\partial x_i} \right) - \frac{2}{3} \delta_{ij} \left( \mu_t \frac{\partial \overline{u_k}}{\partial x_k} + \rho k \right)
\]

Equation (4.23) also introduces \( \mu_t \) which is the turbulent viscosity. This term will be subject to modelling and discussed in subsequent subchapters. The Reynolds averaged momentum equation seen in Equation (4.15) can finally be written as:

\[
    \frac{\partial}{\partial t} (\rho \overline{u_i}) + \frac{\partial}{\partial x_j} (\rho \overline{u_i} \overline{u_j}) = - \frac{\partial p^*}{\partial x_i} + \frac{\partial}{\partial x_j} \left( \mu_{eff} \left( \frac{\partial \overline{u_i}}{\partial x_j} + \frac{\partial \overline{u_j}}{\partial x_i} \right) \right)
\]

\[
    \mu_{eff} = \mu + \mu_t
\]

Firstly, the pressure term is modified to include the two last terms in Equation (4.23):

\[
    p^* = p + \frac{2}{3} \rho k + \frac{2}{3} \mu_{eff} \frac{\partial \overline{u_k}}{\partial x_k}
\]

CFX disregards the last term on the right hand side even though it is not fully correct and equates by default \( p^* \) with \( p \). Secondly, the stress term on the right hand side now have an effective viscosity \( \mu_{eff} \) in which the molecular and turbulent viscosity is included in. For the energy equation, the turbulent flux \( \rho \overline{u_i' h'} \) is modelled in the analogous way with the introduction of the turbulent diffusivity \( \Gamma_t \):

\[
    -\rho \overline{u_i' h'} = \Gamma_t \frac{\partial \overline{h}}{\partial x_j}
\]

The turbulent diffusivity \( \Gamma_t \) is given as \( \Gamma_t = \frac{\mu_t}{0.9} \) by default in CFX. The assumption of a constant relation between the turbulent viscosity and turbulent diffusivity is valid for a Prandtl number (momentum diffusivity over thermal diffusivity) larger than 0.5. In this thesis, the
fluid is air which has a Prandtl number of approximately 0.7 so the constant relation between the turbulent viscosity and turbulent diffusivity is a valid assumption (Schlichting & Gersten, 2006). The energy equation found in Equation (4.17) can, with the Boussinesq assumption, now be written as:

$$\frac{\partial}{\partial t} (\rho H) + \frac{\partial}{\partial x_j} \left( \rho \bar{u}_j H + \Gamma_t \frac{\partial \bar{H}}{\partial x_j} - \lambda \frac{\partial \bar{T}}{\partial x_j} \right) = \frac{\partial \bar{p}}{\partial t} + \frac{\partial}{\partial x_j} (\bar{u}_i ( \bar{F}_{ij} - \rho \bar{u}_i \bar{u}_j ))$$

(4.27)

4.3.4 Two equation models

To solve the Reynolds averaged equations the turbulent viscosity $\mu_t$ must be determined in some manner since it cannot be taken as constant in the flow field.

4.3.4.1 The $k$-$\varepsilon$ model

The $k$ – $\varepsilon$ model is a two equation model which incorporates two additional transport equations, one for the turbulent kinetic energy $k$ and one for the dissipation rate of turbulent kinetic energy $\varepsilon$. With these two quantities, the turbulent viscosity (and the turbulent diffusivity) can be formed as:

$$\mu_T = \rho C_{\mu} k^2 / \varepsilon$$

(4.28)

where $C_{\mu}$ is a model constant. The transport equation from turbulent kinetic energy $k$ and dissipation rate $\varepsilon$ are modelled as (CFX Solver Theory Guide, ANSYS Inc, 2012):

$$\frac{\partial (\rho k)}{\partial t} + \frac{\partial (\rho \bar{u}_j k)}{\partial x_j} = \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_T}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] + P_k - \rho \varepsilon$$

(4.29)

$$\frac{\partial (\rho \varepsilon)}{\partial t} + \frac{\partial (\rho \varepsilon \bar{u}_j)}{\partial x_j} = \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_T}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_j} \right] + \frac{\varepsilon}{k} \left( C_{\varepsilon 1} P_k - C_{\varepsilon 2} \rho \varepsilon \right)$$

(4.30)

The model constants are $C_{\mu} = 0.09, \sigma_k = 1.0, \sigma_\varepsilon = 1.3, C_{\varepsilon 1} = 1.44, C_{\varepsilon 2} = 1.92$. The middle term on the right hand side is the production of turbulence as a result of viscous effects and is modelled as:

$$P_k = \mu_T \left( \frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right) \frac{\partial \bar{u}_i}{\partial x_j} - \frac{2 \partial \bar{u}_k}{\partial x_k} \left( 3 \mu_T \frac{\partial \bar{u}_k}{\partial x_k} + \rho k \right)$$

(4.31)

The $k$ – $\varepsilon$ model is an elegant and simple turbulence model but there are weaknesses. One example is the over prediction of the spreading rate of the round jet. This can be fixed by tweaking the model constants and/or add extra terms. The drawback is that when trying to fix the model for a particular class of flows, the performance for other classes are inferior to the original model formulation (Pope, 2000). Another problem with the $k$ – $\varepsilon$ model is that when approaching the wall, the term $\varepsilon^2/k$ becomes singular since $k$ goes to zero the wall (Johansson & Wallin, 2012). For high Reynolds numbers, this is no problem since the log-law is applicable for $30 < y^+ < 500$ and the velocity, turbulent kinetic energy $k$ and dissipation rate $\varepsilon$ can be modelled with wall functions. For low Reynolds numbers, the log-law does not apply.
and modifications must be made to the transport equations (Versteeg & Malalasekera, 2007). These modified models often require a \( y^+ < 0.2 \) which is very costly and impractical (CFX Solver Theory Guide, ANSYS Inc, 2012).

### 4.3.4.2 The k-\( \omega \) model

This subchapter is based on CFX Solver Theory Guide, ANSYS Inc (2012). There are naturally other ways to form the turbulent diffusivity. One can regard the variable \( \omega \) which is related to the turbulent viscosity \( \mu_T \) as:

\[
\mu_T = \rho \frac{k}{\omega} \tag{4.32}
\]

From Equations (4.28) and (4.32), the relation between \( \omega \) and \( \varepsilon \) can be seen to be:

\[
\omega = \frac{\varepsilon}{C_\mu k} \tag{4.33}
\]

\( \omega \) can be seen as the frequency of the large scale eddies (Johansson & Wallin, 2012). A basic turbulence model based on the transport of \( k \) and \( \omega \) was developed by Wilcox:

\[
\frac{\partial (\rho k)}{\partial t} + \frac{\partial}{\partial x_j} (\rho k \bar{u}_j) = \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_T}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] + P_k - \beta' \rho k \omega \tag{4.34}
\]

\[
\frac{\partial (\rho \omega)}{\partial t} + \frac{\partial}{\partial x_j} (\rho \omega \bar{u}_j) = \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_T}{\sigma_\omega} \right) \frac{\partial \omega}{\partial x_j} \right] + \frac{\omega}{k} P_k - \beta \rho \omega^2 \tag{4.35}
\]

The constants are \( \beta' = 0.09, \alpha = \frac{5}{9}, \beta = 0.075, \sigma_k = 2, \sigma_\omega = 2. \)

The value \( \omega \sim \frac{\varepsilon}{k} \) will approach infinity when going close to wall since \( k \) approaches zero but it’s been seen that this can be avoided by setting the value of \( \omega \) high enough (Johansson & Wallin, 2012). Thus the \( k-\omega \) model is better than the \( k-\varepsilon \) close to the wall. However, in the form developed by Wilcox, the model is too sensitive the conditions of the free stream. Menter developed a hybrid model which uses the \( k-\varepsilon \) model at the free stream and blends to a \( k-\omega \) formulation close to the wall, this is known as the Baseline \( k-\omega \) model. Later on, an addendum by the same author models the effect of the transport of the turbulent shear stress. This model, known as the Shear Stress Transport (SST), has a very good prediction of flow separation. This model is a popular one and is the turbulence model that is used in this thesis.

### 4.3.5 Reynolds Stress Models (RSM) and Large Eddy Simulations (LES)

This subchapter is based on the work of Versteeg & Malalasekera (2007). The two equation models presented previously relies on the Boussinesq assumption. However, flows with large changes in mean strain rate or streamline curvature are not well captured by unmodified turbulence models. Another approach is to calculate the transport equations for each Reynolds stress component. This yields 6 additional partial differential equations (PDEs). Additionally, a transport equation for the dissipation rate \( \varepsilon \) is needed. This approach is general and yields better results for asymmetric channel flow, non-circular duct flow and also flows with curvature. However, it is computationally expensive since it solves 7 additional PDEs. It shares the dissipation rate modelling with the standard \( k-\varepsilon \) model and shows the same deficiencies in certain flow cases.
Large Eddy Simulations (LES) is a rather different approach to the modelling of turbulence. Large turbulent eddies can be seen as rather anisotropic and related to the geometry of the problem whereas the small eddies are more isotropic. Therefore a single model that covers all kinds of flow geometries are quite a challenge to find. LES acknowledges this problem and sets out to calculate the structure of the time-dependent larger eddies but models the smaller eddies which are assumed to be isotropic and thus simple to model. LES is rather expensive as compared to the two-equation models but is has been observed that it might be sufficient with twice the computational power as compared with RSM.

4.4 Numerical Schemes

To solve the transport equations presented previously, a numerical approach must be adopted since there exists no analytical solution for the full Navier-Stokes equations. This subchapter is based on the CFX Solver Theory Guide, ANSYS Inc (2012) since it is the flow solver used in this thesis.

CFX uses a finite volume approach where nodes are used to store variables and a control volume is built up around the node. Furthermore, the governing equations are integrated over the control volumes and gradient and divergence operators are converted to surface integrals through Gauss’ divergence theorem. The volume and surface integrals are then discretized, the details are not discussed here. Several of the terms in the now discretized equation must be evaluated not at the node where all the variables are stored but rather at the surface of the control volume, denoted as integration point, \( ip \). CFX uses shape functions to approximate the values.

The advective term for a quantity \( \phi \) at the integration points can be calculated in several ways. A general form can be seen in Equation (4.36) where different choices of \( \beta \) and \( \nabla \phi \) yield different schemes. The vector from the upwind node to the integration point is given by \( \vec{r} \).

\[
\phi_{ip} = \phi_{\text{upwind}} + \beta \nabla \phi \cdot \Delta \vec{r}
\]  

(4.36)

If \( \beta \) is equal to zero, a first order upwind scheme is obtained. While robust, it will introduce diffusive errors which will smear out the solution. If \( \beta \) is between 0 and 1 and \( \nabla \phi \) is taken as the “average of the adjacent nodal gradients”, the numerically induced diffusivity will be reduced. For \( \beta \) of 1, the scheme is formally second order in space accurate, however dispersive errors might appear which manifest as wiggles. The High Resolution Scheme varies \( \beta \) for each node and tries to keep it as close to 1 as possible and \( \nabla \phi \) is taken from the upwind node only. This is the advective scheme used in this thesis. Diffusion terms and the pressure gradient term are evaluated through a standard finite element treatment and the details will not be treated here. The control volumes are co-located for all quantities, this is known to induce a decoupled pressure field. However, a different discretization (not treated here) is implemented to remedy this. The temporal discretization is given by the backward Euler method. The First Order Backward Euler is fully implicit, bounded, robust and there is no limit in step size, however the first order accuracy will make it prone to diffusive discretization errors which smooth sharp temporal gradients. A higher order discretization is the Second Order Backward Euler which is also implicit, robust and has no step size limit but
may yield numerical oscillations. CFX solves the unsteady equations even for steady state problems where it approaches a steady solution through advancing the approximate solution in time.

CFX is a coupled solver which solves for \( u \) and \( p \) as an equation system and is fully implicit. The coupled solver is more robust, general, simple and efficient. However, it requires large storage space. Segregated solvers use an approximate pressure to solve the momentum equations and then correct the pressure field. This results in a larger number of iterations as compared with the coupled solver. However, segregated solvers do not need to store as much information as coupled solvers.
5 Methodology

5.1 Parameters of study

Binder et al. (2014) investigated experimentally the effect of $L/D$, $w/D$, and pressure ratio on the discharge coefficient and produced a correlation valid for:

$$
\frac{L}{D} \in [0.33; 2.14] \\
\frac{w}{D} \in [0; 0.161] \\
\frac{p_1}{p_2} \in [1.05; 1.8] \quad (5.1)
$$

The goal with the current study is to supplement the work with data from radiused orifices with and without cross flow. Binder et al. (2014) used 1704 data points for the creation of the correlation whereas this study will use around 110 data points due to time constraints combined with the fact that a CFD simulation for one data point takes longer time than its experimental counterpart. The parameters studied and their ranges were:

$$
\frac{L}{D} \in [0.4; 4.0] \\
\frac{r}{D} \in [0; 0.30] \\
\frac{p_1}{p_2} \in [1.05; 1.4] \\
\frac{U}{c_{ax}} \in [0.17; 1.12] \quad (5.2)
$$

This work is divided into two parts. The first part, Study I, concerns flow through orifices without inlet cross flow and the second part, Study II, treats flow through orifices with inlet cross flow. The orifices studied for both parts had a diameter $D$ of 7 mm and their dimensions are presented in Table 5.1 and Table 5.2.

**Table 5.1: Orifice geometry, Study I.**

<table>
<thead>
<tr>
<th>L/D [-]</th>
<th>(r/D)$_1$ [-]</th>
<th>(r/D)$_2$ [-]</th>
<th>(r/D)$_3$ [-]</th>
<th>(r/D)$_4$ [-]</th>
<th>(r/D)$_5$ [-]</th>
<th>(r/D)$_6$ [-]</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.4</td>
<td>0</td>
<td>0.04</td>
<td>0.08</td>
<td>0.16</td>
<td>0.20</td>
<td>-</td>
</tr>
<tr>
<td>1.5</td>
<td>0</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>2.0</td>
<td>0</td>
<td>0.04</td>
<td>0.08</td>
<td>0.16</td>
<td>0.20</td>
<td>0.30</td>
</tr>
<tr>
<td>4.0</td>
<td>0</td>
<td>0.04</td>
<td>0.08</td>
<td>0.16</td>
<td>0.20</td>
<td>0.30</td>
</tr>
</tbody>
</table>
Table 5.2: Orifice geometry, Study II.

<table>
<thead>
<tr>
<th>L/D [-]</th>
<th>(r/D)₁ [-]</th>
<th>(r/D)₂ [-]</th>
<th>(r/D)₃ [-]</th>
<th>(r/D)₄ [-]</th>
<th>(r/D)₅ [-]</th>
</tr>
</thead>
<tbody>
<tr>
<td>2.0</td>
<td>0</td>
<td>0.08</td>
<td>0.16</td>
<td>0.30</td>
<td>0.08</td>
</tr>
<tr>
<td>4.0</td>
<td>0</td>
<td>0.08</td>
<td>0.16</td>
<td>0.30</td>
<td>0.08</td>
</tr>
</tbody>
</table>

The orifice with \( L/D = 1.5 \) in Study I was used for verification and validation purposes only. It can also be seen that for \( L/D = 0.4 \), the largest radius to diameter ratio was not applied due to the fact that such a large radius is rarely used in industrial application for the specific \( L/D \) and was thus disregarded in this work. Due to time constraints Study II contains fewer geometries than Study I, it can be seen for example that no orifices with \( L/D = 0.4 \) were included.

5.2 Geometry

This work is a continuation of the experimental study by Binder et al. (2014) and great effort was taken in ensuring that the computational domain would accurately represent the experimental setup. A schematic diagram of the test rig used by Binder et al. (2014) can be seen in Figure 5.1.

![Figure 5.1: Schematic diagram of test rig (Binder et al., 2014).](image)

A pressurized air system was connected to the flow control valve from which the flow was regulated. The Coriolis flow meter measured the mass flow of the passing air. The air then entered into a pressure vessel and finally emptied out into the ambient environment through an orifice due to the pressure difference. The orifice diameter \( D \) was 7 mm the diameter of the pressure vessel was 330.2 mm or approximately 47\( D \). To capture the same flow physics in the CFD calculation, the geometry modelled included the pressure vessel, the orifice and the ambient environment. It is however infeasible to model the whole pressure vessel due to the sheer amount of computational cells needed so a simplification is needed. Also, a challenge arises in modelling the ambient environment. How should it be modelled so physically meaningful boundary conditions can be applied?

Binder et al. (2014) determined the pressure vessel to be large enough that the dynamic to total pressure ratio did not exceed 0.0185%. The flow in the vessel can be considered to be
quiescent in large parts of the pressure vessel, save the region closest to the orifice. The modelling of the pressure vessel can then be restricted to the region closest to the orifice. From the orifice’s point of view, the flow approaches from all directions as the diameter of the pressure vessel is much larger than the diameter of the orifice. To model this, a hemisphere can be used to represent the pressure vessel since a boundary condition with a normal velocity direction can be set on the curved hemisphere surface. The orifice is geometrically a cylinder and modelled as such. The ambient environment was also modelled as a hemisphere. From the orifice exit, the radial distance to the hemisphere boundary is equal for all points on the boundary: Therefore a boundary condition on the curved surface of the hemisphere applies equally on the orifice and is sufficient for modelling the ambient environment.

The hemisphere used for modelling the pressure vessel can be seen to the left in Figure 5.2 (a) and is much smaller than the hemisphere used for modelling the ambient environment which can be seen to the right in the same figure. The reason for the large hemisphere representing the ambient environment is that the jet emerging into the ambient environment can potentially cause convergence problems if the gradients are still strong at the boundary, therefore a larger geometry was used to reduce the risk of large gradients at the boundary.

![Figure 5.2: Geometry of computational domain L/D = 4.0, r/D = 0.08. (a) Study I. (b) Study II.](image)

For Study II, where cross-flow is imposed on the inlet, the geometries from Study I could be used with a slight modification. To account for cross-flow, an inlet box has to be attached to the inlet hemisphere. The situation is now quite different for the pressure vessel and it can be seen as a duct with an orifice in the side of the duct. An example of the geometry for Study II can be seen in the Figure 5.2 (b). The hemisphere representing the pressure vessel is henceforth denoted as Hemisphere I, the hemisphere representing the ambient environment as Hemisphere II and the cube attached to Hemisphere I as Cross-flow cube. The dimensions can be found in Table 5.3.

Table 5.3: Dimensions of parts in the computational domain.

<table>
<thead>
<tr>
<th>L/D = 0.4</th>
<th>(r/D)₁ [-]</th>
<th>(r/D)₂ [-]</th>
<th>(r/D)₃ [-]</th>
<th>(r/D)₄ [-]</th>
<th>(r/D)₅ [-]</th>
<th>(r/D)₆ [-]</th>
</tr>
</thead>
<tbody>
<tr>
<td>Hemisphere I diameter</td>
<td>14D</td>
<td>10D</td>
<td>10D</td>
<td>10D</td>
<td>10D</td>
<td>-</td>
</tr>
<tr>
<td>Hemisphere II diameter</td>
<td>20D</td>
<td>20D</td>
<td>20D</td>
<td>20D</td>
<td>20D</td>
<td>-</td>
</tr>
<tr>
<td>Cross-flow cube length</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>L/D = 1.5</td>
<td>(r/D)₁ [-]</td>
<td>(r/D)₂ [-]</td>
<td>(r/D)₃ [-]</td>
<td>(r/D)₄ [-]</td>
<td>(r/D)₅ [-]</td>
<td>(r/D)₆ [-]</td>
</tr>
<tr>
<td>-----------</td>
<td>------------</td>
<td>------------</td>
<td>------------</td>
<td>------------</td>
<td>------------</td>
<td>------------</td>
</tr>
</tbody>
</table>
For sharp-edged orifices, \((r/D)_1\), the diameter of Hemisphere I can be seen to be larger than for other geometries, this is due to the fact that it was discovered later on that reducing the diameter of Hemisphere I from 14D to 10D did not alter the mass flow significantly. Subsequent cases were done with a reduced diameter to save computational time. In the case with inlet cross-flow, the Cross-flow cube length is 12D for all geometries, however a validation study was made with a length of 8D instead for \(L/D = 2.0\), \((r/D)_1\) in order to better replicate the experimental study.

### 5.3 Computational mesh

An adequate computational mesh is crucial when performing CFD calculations and a large portion of the time of the work was spent on the creation of the computational meshes. A mesh was made for each geometry in ICEM 14.5.7, however they are quite similar and the basic structure of the mesh can be divided into three categories. A detail of the mesh element counts can be found in Appendix A.

#### 5.3.1 Mesh type I, sharp-edged orifices

For all sharp-edged orifices, i.e. \(r/D = 0\), the final mesh is composed of three different meshes connected to each other with a General Grid Interface (GGI). The inner structure was meshed with a structured hexahedral mesh and was composed of a cube connected to an orifice which in turn was connected to a second cube. The reason for this strategy was that a structured hexahedral mesh was desired since it offers more control when meshing. However this was not realizable for the full geometry and the hemispheres were meshed with unstructured tetrahedrals.
5.3.2 Mesh type II, radiused orifices

The structured mesh in radiused orifices could, due the nature of the geometry, be extended to include Hemisphere I. This resulted in a structured hexahedral mesh composed of Hemisphere I, the orifice and a cube, see Figure 5.5 (a), and an unstructured tetrahedral mesh identical to the type I mesh seen in Figure 5.4 (a). The meshes were also connected through a GGI and the final form can be observed in Figure 5.5 (b).
5.3.3 Mesh type III, cross-flow

To accommodate for inlet cross-flow, a cube had to be attached to the Hemisphere I. For sharp-edged orifices (mesh type I), the structured mesh seen in Figure 5.3 (a) had already the shape of a cube so the only modification was to enlarge the cube seen to left in the same figure and disregard the Hemisphere I mesh seen in Figure 5.3 (b). An example of the structured hexahedral mesh for sharp-edged orifices adapted for cross-flow can be seen in Figure 5.6 (a), also the final mesh can be seen in Figure 5.6 (b).

Figure 5.6: L/D = 4.0, r/D = 0, cross-flow. (a) Structured hexahedral mesh adapted for cross-flow. (b) Final mesh.

For radiused orifices (mesh type II), an unstructured tetrahedral mesh was applied to a cube geometry and then connected to Hemisphere I through a GGI. The unstructured tetrahedral mesh can be seen in Figure 5.7.

Figure 5.7: L/D = 4.0, r/D = 0.08, cross-flow. Unstructured tetrahedral mesh, Cross-flow cube.

5.4 General Parameters

The commercial CFD solver ANSYS CFX 14.5 was used. A RANS-based turbulence model, k-ω SST was used to account for turbulent flow. The fluid was air treated as an ideal gas to account for compressibility. The total energy equation was used since the flow regime is in most cases above the compressible limit (M > 0.3). The viscous work term was disregarded since it is “negligible for most flows” and the turbulent Prandtl number, which states the relation between the turbulent viscosity µt and the turbulent diffusivity Γt, was set to its default value of 0.9 (CFX-Solver Theory Guide, ANSYS Inc, 2012).

The advection scheme was set to “High resolution”. This option utilizes a blend factor and the formal order of accuracy is aimed to be as close to second order as possible. Similarly, the turbulent scheme was also set to “High resolution” which uses the same advection scheme.
To treat the turbulent boundary layer an automatic wall function was used. It switches from a low Reynolds formulation for \( y^+ < 2 \) to a wall function for higher values of \( y^+ \) (CFX Solver Modeling Guide, ANSYS Inc, 2012). Due to the separation in the orifice, it was believed that a \( y^+ \approx 1 \) should hold at the orifice wall to yield more accurate results and the meshes was created accordingly. In the regions where no flow separation was expected, the meshes were created with the criterion of \( y^+ \leq 30 \). For certain cross-flow simulations, it was revealed that \( y^+ = 3 \) at the inlet region in the orifice but this issue was not addressed due to time constraints.

Steady simulations were aimed for but for certain flow cases with short orifices \( (L/D = 0.4) \), convergence could not be reached and oscillating residuals were present which hints an unsteady behaviour. Transient simulations were performed and the desired quantities were time averaged.

The convergence criterion was set as the RMS values of the residual to be less than \( 1 \cdot 10^{-5} \) for steady state simulations and \( 1 \cdot 10^{-4} \) for transient simulations. Two additional tools were used to determine if convergence has been achieved. Firstly, monitor points were used in different regions of the computational domain to examine if the quantities had become steady (or oscillating with finite and consistent amplitude for transient simulations). The quantities monitored were velocity, pressure and turbulent kinetic energy. Secondly, the domain imbalance of mass, momentum and energy were taken into consideration. The highest imbalance was found to be approximately 0.3 % for the energy, this can be taken as acceptable.

A summary of the most important solver settings can be seen in Table 5.4.

Table 5.4: Solver settings.

<table>
<thead>
<tr>
<th>Fluid</th>
<th>Air ideal gas</th>
</tr>
</thead>
<tbody>
<tr>
<td>Turbulence model</td>
<td>K-( \omega ) SST (beta reattachment modification for ( L/D &gt; 0.4 ))</td>
</tr>
<tr>
<td>Wall function</td>
<td>Automatic</td>
</tr>
<tr>
<td>Heat transfer</td>
<td>Total energy (viscous work disregarded)</td>
</tr>
<tr>
<td>Turbulent Prandtl number</td>
<td>0.9</td>
</tr>
<tr>
<td>Transient scheme</td>
<td>Second order backward Euler</td>
</tr>
<tr>
<td>Advection scheme</td>
<td>High resolution</td>
</tr>
<tr>
<td>Turbulent scheme</td>
<td>High resolution</td>
</tr>
</tbody>
</table>
| Convergence criterion RMS | Steady: \( 1 \cdot 10^{-5} \)  
                         | Transient: \( 1 \cdot 10^{-4} \)         |

5.5 Boundary conditions

The boundary conditions will differ between the cases with and without cross-flow and will be treated in the following subchapters.

5.5.1 No cross-flow

For flow cases without cross-flow, the boundary conditions should represent the experimental study by Binder et al. (2014). This calls for a pressure inlet/pressure outlet boundary condition. Hemisphere I, which represents the pressure vessel, had a boundary condition of type “Inlet” on the curved surface with a total pressure equal to the pressure in the vessel specified with velocity entering normally through the boundary. The total pressure here is equivalent to the static pressure due to the quiescent air which is also the situation in the
pressure vessel. Hemisphere II, representing the ambient environment, had a boundary condition of type “Opening” with a specified opening pressure equal to the ambient pressure. The “Opening” boundary condition allows the flow to re-enter the domain if required (CFX Solver Modeling Guide, ANSYS Inc, 2012). The boundary condition “Outlet”, which does not allow flow to re-enter the domain was also evaluated but no convergence could be reached and backflow was present. As a consequence the decision was made to use the “Opening” type. The walls were modelled as smooth adiabatic walls with a no-slip condition. The pressure vessel was large and the flow velocity is near zero in it, thus the turbulence intensity level $Tu$ was set to 1%.

CFX offers the possibility of having two different meshes connected to each other through a General Grid Interface (GGI). They do not have to match each other and can even be of different types. All the connections between the different meshes explained in subchapters 5.3.1-5.3.3 are made by a GGI. A schematic of the boundary conditions can be seen in Figure 5.8. Note that for sharp-edged orifices, an additional GGI is inserted between Hemisphere I and the cube.

![Figure 5.8: Boundary conditions. L/D = 4.0, r/D = 0.08.](image)

The boundary condition values were taken from experimental data and varied from case to case. A reference pressure was set in the solver and the pressures defined on the boundaries are gauge pressures. A summary of the boundary conditions for this subchapter can be found in Table 5.5.

**Table 5.5: Boundary conditions, no cross-flow.**

<table>
<thead>
<tr>
<th>Reference pressure</th>
<th>≈100000 Pa</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Inlet</strong></td>
<td></td>
</tr>
<tr>
<td>Total pressure (gauge)</td>
<td>≈1000-70000 Pa</td>
</tr>
<tr>
<td>Total temperature</td>
<td>≈292-296 K</td>
</tr>
<tr>
<td>Turbulence intensity</td>
<td>1%</td>
</tr>
<tr>
<td>Flow direction</td>
<td>Normal to boundary condition</td>
</tr>
</tbody>
</table>
### Outlet
<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Opening pressure (gauge)</td>
<td>0 Pa</td>
</tr>
<tr>
<td>Opening temperature</td>
<td>292.15-293.15 K</td>
</tr>
<tr>
<td>Turbulence intensity</td>
<td>1%</td>
</tr>
<tr>
<td>Flow direction</td>
<td>Normal to boundary condition</td>
</tr>
</tbody>
</table>

### Wall
<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mass and momentum</td>
<td>No slip</td>
</tr>
<tr>
<td>Heat transfer</td>
<td>Adiabatic</td>
</tr>
<tr>
<td>Wall roughness</td>
<td>Smooth</td>
</tr>
</tbody>
</table>

### GGI
<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Boundary type</td>
<td>Interface</td>
</tr>
<tr>
<td>Mass and momentum</td>
<td>Conservative interface flux</td>
</tr>
<tr>
<td>Turbulence</td>
<td>Conservative interface flux</td>
</tr>
<tr>
<td>Heat transfer</td>
<td>Conservative interface flux</td>
</tr>
</tbody>
</table>

#### 5.5.2 Cross-flow

The boundary conditions changes when studying cross-flow. A cross-flow velocity has to be imposed on the inlet and a supplementary outlet has to be defined. A schematic of the boundary conditions can be seen for \( L/D = 4.0 \), \( r/D = 0.08 \) in Figure 5.9.

![Figure 5.9: Boundary conditions, cross-flow.](image)

The pressure ratio across the orifice was desired to be 1.1 which corresponds to engine-like conditions. This was obtained by setting a static pressure condition on the outlet to 10132.5 Pa (g). The inlet boundary condition was set as normal speed. The author believes that total pressure could have been used as well. The study of Binder et al. (2014) did not include cross-flow so the reference pressure, total temperature at the inlet and opening temperature are set as constant values for all cases. The remaining boundaries (walls and GGI) were identical to the ones without cross-flow. A summary of the boundary conditions that have been altered is presented in Table 5.6.
Table 5.6: Boundary conditions, cross-flow.

<p>| | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Reference pressure</strong></td>
<td>101325 Pa</td>
</tr>
<tr>
<td><strong>Inlet</strong></td>
<td></td>
</tr>
<tr>
<td>Normal speed</td>
<td>22.2-150.0 m/s</td>
</tr>
<tr>
<td>Total temperature</td>
<td>294 K</td>
</tr>
<tr>
<td>Turbulence intensity</td>
<td>1 %</td>
</tr>
<tr>
<td><strong>Outlet</strong></td>
<td></td>
</tr>
<tr>
<td>Average static pressure (gauge)</td>
<td>10132.5 Pa</td>
</tr>
<tr>
<td><strong>Opening</strong></td>
<td></td>
</tr>
<tr>
<td>Opening pressure (gauge)</td>
<td>0 Pa</td>
</tr>
<tr>
<td>Opening temperature</td>
<td>293.15 K</td>
</tr>
<tr>
<td>Turbulence intensity</td>
<td>1 %</td>
</tr>
</tbody>
</table>

5.6 Mesh verification

A computational mesh should ideally be able to accurately model the flow situation as cost-efficiently as possible. It is therefore necessary to verify that the mesh used is not too coarse for the application, since it will yield inaccurate results but also not too fine since the additional computational effort will not yield any different result. Due to time constraints, mesh verification was only performed for flow cases without cross-flow. Preferably, mesh verification should have been carried out for flow cases with cross-flow since it represents different flow physics.

5.6.1 No cross-flow

Two geometries were chosen with each representing its mesh type.

5.6.1.1 L/D = 1.5, r/D = 0

A mesh verification was performed on a sharp-edged orifice with L/D = 1.5, the meshes are of type I seen in subchapter 5.3.1. Two different pressure ratios, 1.1 and 1.5, each representing an extreme of the range of pressure ratios, were investigated. The main variable considered is the mass flow since it has a crucial part in determining the discharge coefficient.

Four different meshes, numbered 1-4 were generated. All meshes are based on Mesh 3 and scaled up or down. The scaling was done in two different ways. For the structured hexahedral mesh, a refinement factor was specified. A refinement factor less than one is equivalent with a coarsening of the mesh, whereas a refinement factor larger than one corresponds to a refinement of the mesh. However, the refinement of the unstructured tetrahedral mesh was performed through a scale factor. A scale factor less than one yields a refinement of the mesh, whereas a scale factor larger than one results in a coarsening of the mesh. It might also be worth to note that while the mesh was scaled the initial spacing of the first element was held constant to retain the same desired value of $y^+$ for the structured hexahedral mesh. A summary of the details can be seen in Table 5.7.
Table 5.7: Mesh verification. L/D = 1.5, r/D = 0, mesh parameters.

<table>
<thead>
<tr>
<th>Mesh</th>
<th>Elements</th>
<th>Elements across the orifice</th>
<th>Refinement factor hexahedral mesh</th>
<th>Global element scale factor tetrahedral mesh</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1452755</td>
<td>87</td>
<td>0.6</td>
<td>1.4</td>
</tr>
<tr>
<td>2</td>
<td>3249319</td>
<td>115</td>
<td>0.8</td>
<td>1.2</td>
</tr>
<tr>
<td>3</td>
<td>5940630</td>
<td>140</td>
<td>1</td>
<td>1</td>
</tr>
<tr>
<td>4</td>
<td>10551087</td>
<td>168</td>
<td>1.2</td>
<td>0.8</td>
</tr>
</tbody>
</table>

For each mesh, the refinement factor and scale factor does not correspond one-to-one. An estimation of how consistent the refining and coarsening is between hexahedral and tetrahedral elements is given by taking the average volume of an element and compare it with the average volume of the element in Mesh 3. From this, it can be seen in Table 5.8 that the refinement/coarsening gives a consistent ratio between both element types except for Mesh 1 where it can be seen that the element size ratio for the hexahedral elements is roughly 60% larger than the element size ratio for the tetrahedral elements.

Table 5.8: Average element volume comparison normalized with the element volume of Mesh 3.

<table>
<thead>
<tr>
<th>Mesh</th>
<th>Avg. hexahedral volume ratio [-]</th>
<th>Avg. tetrahedral volume ratio [-]</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>4.3690</td>
<td>2.7425</td>
</tr>
<tr>
<td>2</td>
<td>1.8439</td>
<td>1.7184</td>
</tr>
<tr>
<td>3</td>
<td>1</td>
<td>1</td>
</tr>
<tr>
<td>4</td>
<td>0.5699</td>
<td>0.5085</td>
</tr>
</tbody>
</table>

The results from the mesh verification for a pressure ratio $p_1/p_2 = 1.1$ are presented in Table 5.9. Each result is compared with the finest mesh, Mesh 4.

Table 5.9: Mesh verification. L/D = 1.5, r/D = 0, results.

<table>
<thead>
<tr>
<th>Mesh</th>
<th>Pressure ratio [-]</th>
<th>Mass flow [g/s]</th>
<th>Relative difference with mesh 4 [%]</th>
<th>Separation region length [mm]</th>
<th>Relative difference with mesh 4 [%]</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1.1</td>
<td>4.602</td>
<td>0.18</td>
<td>5.49</td>
<td>6.3</td>
</tr>
<tr>
<td>2</td>
<td>1.1</td>
<td>4.600</td>
<td>0.13</td>
<td>5.65</td>
<td>3.58</td>
</tr>
<tr>
<td>3</td>
<td>1.1</td>
<td>4.596</td>
<td>0.05</td>
<td>5.86</td>
<td>0</td>
</tr>
<tr>
<td>4</td>
<td>1.1</td>
<td>4.594</td>
<td>-</td>
<td>5.86</td>
<td>-</td>
</tr>
</tbody>
</table>

It can be seen that the difference in mass flow is very low, with the highest difference of 0.18% for the coarsest mesh. However, the separation region lengths in the orifice (extracted by sampling along the orifice length for all meshes to enable comparison) vary noticeably with the largest difference being 6.3% for the coarsest mesh. However, the mass flow is the main quantity of concern so although the separation region lengths differ, Mesh 1 is sufficient for future studies. Furthermore, the separation region for Mesh 1 and Mesh 4 is consistent in shape which can be seen in Figure 5.10 (a) and (b) by plotting the negative axial velocities.
Figure 5.10: Separation region illustrated by contour plots of negative axial velocity. (a) Mesh 1. (b) Mesh 4.

It is also important to ensure that the profiles of certain quantities are consistent between the meshes. In this verification, the axial velocity profiles and turbulent kinetic energy (TKE) profiles were taken from lines across the orifice as illustrated in Figure 5.11 for the four meshes. Only the closest and farthest lines will be treated.

Figure 5.11: Lines across the orifice.

The values are given in a non-dimensionalized form and are based on the Cartesian coordinate system:
- The axial distance is denoted as $x/L$ where $L$ is the length of the orifice.
- The distance across the orifice, in this case $y$ and $z$, are translated and normalized with the orifice diameter $D$ and denoted as $y/D$ and $z/D$ respectively.
- Furthermore, the axial velocity and turbulent kinetic energy are normalized with the largest values obtained in Mesh 4.

Firstly, the closest line to the orifice inlet at $x/L=0.095$, is examined. It can be observed in Figure 5.12 that the axial velocity profiles are similar in shape and in magnitude; however discrepancies can be observed at the extremes where the profiles do not match well. Figure 5.12 also offers an insight of the orifice flow field. There is a negative velocity region close to the wall due to the separation region. The axial velocity then increases drastically when moving further away from the wall, this is due to flow acceleration due to the nozzle formed by the separated region. The free stream velocity is seen at $y/D = 0.5$, i.e. in the middle of the orifice, to be around 80% of the maximum velocity.
It is observed in Figure 5.13 that the relative difference in turbulent kinetic energy between Mesh 1 and Mesh 4, i.e. the coarsest and the finest mesh is quite elevated at the separation region, a comprehensive explanation of this phenomenon cannot be presented but it is evident that mesh size affect the turbulent kinetic energy. Also, the turbulent kinetic energy can be observed to be largest close to the wall and rather low in the free stream.

**Figure 5.12: Axial velocity profile across the orifice at x/L = 0.095.**

**Figure 5.13: Turbulent kinetic energy profile across the orifice at x/L = 0.095.**
Secondly, it is interesting to look at a line outside the separation region. The farthest line from the orifice inlet is located at $x/L = 0.952$. The axial velocity profiles can be regarded in Figure 5.14 it is concluded that the velocity profiles are virtually identical to each other. Furthermore, it can be noted that the turbulent velocity profiles in Figure 5.15 are quite similar to each other to a larger extent than in the separation region seen in Figure 5.13. The axial velocity and
TKE profiles were also evaluated for \( x/L = 0.476 \) but due to similarity with the profiles at \( x/L = 0.952 \), the analysis is omitted and the figures can be found in Appendix B. In addition to visually inspecting the axial velocity and TKE profiles, the relative difference of the quantities between Mesh 1 and Mesh 4 can be extracted and is presented in Table 5.10 for \( x/L = 0.095 \). The data was extracted through interpolation to enable the comparison.

**Table 5.10: Relative difference between Mesh 1 and Mesh 4 in axial velocity and TKE at \( x/L = 0.095 \).**

<table>
<thead>
<tr>
<th>Location</th>
<th>( y/D ) [-]</th>
<th>Rel. difference axial velocity [%]</th>
<th>Rel. difference TKE [%]</th>
</tr>
</thead>
<tbody>
<tr>
<td>Shear layer</td>
<td>0.0142</td>
<td>6.95</td>
<td>64.57</td>
</tr>
<tr>
<td>Centre line</td>
<td>0.5</td>
<td>0.87</td>
<td>15.35</td>
</tr>
</tbody>
</table>

It can be concluded that the relative differences are drastically high, especially at the shear layer. Thus, for these quantities, it can be seen that the coarsest mesh, Mesh 1, does not produce the same results as the finest mesh, Mesh 4. However, the important quantity here is the mass flow and it can be concluded from the observations made in this subchapter that although the axial velocity profiles and TKE profiles do not match exactly, they have a similar shape. This was also observed for the shape of the separation region. Lastly, to ensure that the flow is symmetric in the orifice, the axial velocity is plotted for two perpendicular lines across the orifice for Mesh 1 at \( x/L = 0.095 \). Visual inspection of Figure 5.16 shows that the axial velocities are indeed identical on the two perpendicular lines.

![Figure 5.16: Axial velocity profile across the orifice at \( x/L = 0.095 \), Mesh 1.](image)

From the analysis presented it can then be concluded that Mesh 1, i.e. the coarsest mesh, gives a good performance for the task at hand and its parameters (growth rate, max size) were carried over to the mesh of other sharp-edged geometries.
From this small study, it was seen that the initial mesh created, Mesh 3 was too fine for the task and the same results could be achieved with a mesh with four times less elements.

A mesh verification study was also run for a pressure ratio of 1.5. However, for this pressure ratio, there was evidence of supersonic regions in the orifice as seen in Figure 5.17. The contour plot shows the Mach number and it can be seen that the Mach number exceeds unity at the orifice inlet. Transitions from supersonic velocities to subsonic velocities involve shock waves which require extensive work with the mesh to capture the shock properly. Moreover, the solver might not reach converge as easily as in subsonic flow cases (CFX Solver Modeling Guide, ANSYS Inc, 2012). Thus, the mesh verification for a pressure ratio of 1.5 cannot be taken as valid even though convergence was reached for all four meshes. It gives however an estimate of where supersonic flow starts to appear. Based on this, the maximum pressure ratio was limited to 1.4 for further studies. However, one validation case for a short orifice with \( L/D = 0.4 \) had a pressure ratio up to 1.7 due to the fact the shorter orifices does not choke as early as longer orifices as discussed in subchapter 3.3, and thus supersonic regions are likely to occur later.

![Contour plot of Mach number for Mesh 4, pressure ratio 1.5](image)

**Figure 5.17: Contour plot of Mach number for Mesh 4, pressure ratio 1.5**

### 5.6.1.2 \( L/D = 0.4, r/D = 0.04 \)

The second mesh verification was performed for a short orifice with small inlet radiusing. This mesh represents a Type II mesh as seen in subchapter 5.3.2. Only one pressure ratio which was \( p_1/p_2 = 1.2 \) was utilised which is slightly higher than in the previous subchapter. Three different meshes, numbered 5-7, were created in the same fashion as for the previous mesh verification and are presented in Table 5.11.

**Table 5.11: Mesh verification. \( L/D = 0.4, r/D = 0.04, \) mesh parameters.**

<table>
<thead>
<tr>
<th>Mesh</th>
<th>Elements</th>
<th>Elements across the orifice</th>
<th>Refinement factor hexahedral mesh</th>
<th>Global element scale factor tetrahedral mesh</th>
</tr>
</thead>
<tbody>
<tr>
<td>5</td>
<td>1074824</td>
<td>107</td>
<td>0.8</td>
<td>1.2</td>
</tr>
<tr>
<td>6</td>
<td>1935475</td>
<td>130</td>
<td>1</td>
<td>1</td>
</tr>
<tr>
<td>7</td>
<td>3443258</td>
<td>156</td>
<td>1.2</td>
<td>0.8</td>
</tr>
</tbody>
</table>
The results from the mesh verification are presented in Table 5.12. The absence of the separation region length column is explained by the fact that the flow does not reattach to the orifice for the current flow case which can be seen in Figure 5.18.

**Table 5.12: Mesh verification. L/D = 0.4, r/D = 0.04, results.**

<table>
<thead>
<tr>
<th>Mesh</th>
<th>Pressure ratio [-]</th>
<th>Mass flow [g/s]</th>
<th>Relative difference with mesh 7 [%]</th>
</tr>
</thead>
<tbody>
<tr>
<td>5</td>
<td>1.2</td>
<td>6.54512</td>
<td>0.31</td>
</tr>
<tr>
<td>6</td>
<td>1.2</td>
<td>6.56538</td>
<td>0.0053</td>
</tr>
<tr>
<td>7</td>
<td>1.2</td>
<td>6.56573</td>
<td>-</td>
</tr>
</tbody>
</table>

*Figure 5.18: Separation region illustrated by negative axial velocity contours. L/D = 0.4, r/D = 0.04, Mesh 5.*

It can be deduced from Table 5.12 that Mesh 5, having three times fewer elements than Mesh 7 gives a mass flow that differs only 0.3 % from the latter mesh. However, as in the previous case, the axial velocity and TKE profiles should be compared to ensure that they are consistent before coming to a conclusion. The profiles were evaluated along three different lines but only the line closest to the orifice inlet located at x/L = 0.4 will be presented here due to similarity between profiles. The profiles at x/L = 0.6 and 0.8 can be found in Appendix B. The axial velocity profiles in Figure 5.19 are virtually on top of each other even though it is hinted at there exists a difference at the highest axial velocity.

The TKE profiles observed in Figure 5.20 are in fairly good agreement apart from the fact that there exists a discrepancy between Mesh 5 and Mesh 7 at the peak values of TKE. This is consistent with the observation seen in Figure 5.13, however the coarsest mesh here gives lower values whereas the coarsest mesh in subchapter 5.6.1.1 gives higher values.
Figure 5.19: Axial velocity profile across the orifice at $x/L = 0.4$.

Figure 5.20: Turbulent kinetic energy profile across the orifice at $x/L = 0.4$.

The relative difference between Mesh 5 and Mesh 7 for axial velocity and TKE are, similarly to the previous section, compared at the shear layer and the centre line for $x/L = 0.4$. The data are sampled along the line to enable comparison between the two meshes which have different amount of elements. Table 5.13 shows that there exist a difference between the meshes but the difference is not as dramatic as the difference seen in Table 5.10.
Table 5.13: Relative difference between Mesh 5 and Mesh 7 in axial velocity and TKE at x/L = 0.4.

<table>
<thead>
<tr>
<th>Location</th>
<th>y/D [-]</th>
<th>Rel. difference axial velocity [%]</th>
<th>Rel. difference TKE [%]</th>
</tr>
</thead>
<tbody>
<tr>
<td>Shear layer</td>
<td>0.01</td>
<td>1.21</td>
<td>4.76</td>
</tr>
<tr>
<td>Centre line</td>
<td>0.5</td>
<td>0.30</td>
<td>2.45</td>
</tr>
</tbody>
</table>

Finally, the axial velocity of two perpendicular lines at x/L = 0.4 in Mesh 5 are plotted in Figure 5.21. The axial velocity profiles are identical but the sharp-eyed can observe that the mesh is slightly skewed in the free stream since the location of the data points do not match on two occasions. However, since the profiles otherwise are identical, this skewness is not believed to affect the solution noticeably. From the analysis in this section, it can be concluded that the coarsest mesh, Mesh 5, should be sufficient for the current task. Its parameters (growth ratio, max size) were subsequently carried over to all radiused orifices.

![Figure 5.21: Axial velocity profiles across the orifice at x/L = 0.4, Mesh 5.](image)

5.6.2 Cross-flow

Due to time constraints, no mesh verification could be performed for cross-flow cases. However, the meshes used are partly from the flow cases without cross-flow which were verified in subchapter 5.6.1.

5.7 Solver validation

One of the questions that are always present in CFD calculations is: How do we know that the calculations gives answers that are in agreement with reality, i.e. how much confidence can we have in our CFD results? Validation is the process of comparing the results given by CFD calculations with results obtained experimentally. In this section, validation will be made against data from Binder (2013) for flow cases without cross-flow but also with data from
Hüning (2012) for flow cases with cross-flow. However, before validating the CFD calculations, a small study was made to compare difference turbulence models with data from Binder (2013).

5.7.1 Turbulence models

An overview of the flow cases and the turbulence models studied can be seen in Table 5.14 and Table 5.15. Note that this study was made for cases without cross-flow only. The model of choice up to this point has been the k-ω SST with beta reattachment option but this study was made to benchmark several turbulence models and compare them with experimental data in order to decide which model best suits the needs.

Table 5.14: Geometrical and physical parameters for turbulence model study.

<table>
<thead>
<tr>
<th>L/D [-]</th>
<th>r/D [-]</th>
<th>p1/p2 [-]</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.4</td>
<td>0</td>
<td>1.2</td>
</tr>
<tr>
<td>1.5</td>
<td>0</td>
<td>1.2</td>
</tr>
</tbody>
</table>

The two geometries were chosen since they represent two different flow regimes, the flow in the short orifice will not reattach to the orifice wall whereas the flow in the long orifice will reattach as already seen in the mesh verification study for L/D = 1.5.

Table 5.15: Overview of studied turbulence models.

<table>
<thead>
<tr>
<th>Turbulence model</th>
<th>Option 1</th>
<th>Option 2</th>
</tr>
</thead>
<tbody>
<tr>
<td>k-ω SST</td>
<td></td>
<td></td>
</tr>
<tr>
<td>k-ω SST</td>
<td>Reattachment modification (beta)</td>
<td></td>
</tr>
<tr>
<td>k-ω SST</td>
<td>Reattachment modification (beta)</td>
<td>Curvature correction</td>
</tr>
<tr>
<td>Spalart-Allmaras (beta)</td>
<td></td>
<td></td>
</tr>
<tr>
<td>RNG k-ε</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

The k-ω SST model is explained in section 4.3.4.2. The option “Reattachment modification” is a beta function that is available in CFX only. It has been observed that while most RANS model accurately predicts the separation point, they overpredict the length of the separation region. This model seeks out to correct this by adding an extra production term for TKE at the separated shear layer and the results are in better agreement with experiments compared to the standard k-ω SST model. However, the effects of the model have been shown to vanish when the grid is refined and it is therefore in a beta stage (Lechner et al., 2010). The inlet of the orifice will experience separation and reattachment, therefore this option was included in the study. The curvature correction option tries to correct the fact that two-equation models are insensitive to streamline curvature by adding a modification to the TKE production term (CFX-Solver Theory Guide, ANSYS Inc, 2012). This option was chosen since the flow at the inlet of the orifice experience a curvature when entering the orifice. The Spalart-Allmaras model is a one equation model also in beta stage in CFX 14.5, a general description of it can be found in FLUENT Theory Guide 14.5 (ANSYS Inc, 2012). It is originally designed for wall-bounded flows with an application in aerospace. Relatively large errors are expected for certain shear flows and it is not calibrated for general industrial application. It was chosen since it is a fairly common model used. The RNG k-ε model is a two equation model which
has minor changes from the standard k-ε model and the details are treated in CFX-Solver Modeling Guide (ANSYS Inc, 2012).

From Table 5.16, it can be concluded that the Spalart-Allmaras model has quite good performance for both cases. However, there exists a reason to why this model is not available generally (even though the author has yet to find it) and it has its weaknesses as stated previously. In the same table it can be seen that the standard k-ω SST model has the best performance for a short orifice with $L/D = 0.4$. For a long orifice, $L/D = 1.5$, it can be observed that the k-ω SST with reattachment modification and curvature correction performs best. The RNG k-ε model did not converge for the case with $L/D = 0.4$ and is thus not included in the comparison.

**Table 5.16: Relative difference in mass flow with Binder (2013).**

<table>
<thead>
<tr>
<th>Turbulence model</th>
<th>Relative difference of mass flow with Binder (2013). [%]</th>
</tr>
</thead>
<tbody>
<tr>
<td>k-ω SST</td>
<td>0.72</td>
</tr>
<tr>
<td>k-ω SST reattachment modification (beta)</td>
<td>1.94</td>
</tr>
<tr>
<td>k-ω SST reattachment modification (beta), curvature correction</td>
<td>4.82</td>
</tr>
<tr>
<td>Spalart-Allmaras (beta)</td>
<td>0.70</td>
</tr>
</tbody>
</table>

The mesh verification study in section 5.6.1.1, i.e. $L/D = 1.5$, was performed prior to this study with the k-ω SST with reattachment modification turbulence model. It is observed that this option gives a slightly larger difference. For the sake of consistency, it was decided to continue with this turbulence model for long orifices i.e. orifices with $L/D > 0.4$. On the other hand, for orifices with $L/D = 0.4$, the decision was made that the turbulence model used would be the standard k-ω SST model. The mesh verification in section 5.6.1.2 was done after this study and thus the standard k-ω SST model was used there.

To summarize, for short orifices with $L/D = 0.4$, the k-ω SST model was used whereas for longer orifices with $L/D > 0.4$, the k-ω SST model with reattachment option was used. It can be discussed which model would perform better for short orifices, $L/D = 0.4$, with a large radiusing since the flow will reattach. Would the reattachment modification give more accurate results as it does with $L/D = 1.5$ which has reattachment? It is however impossible to deduce since there is no data to validate against for this case.
5.7.2 Model validation

Three validation cases were made, each chosen to represent its problem class. The first case was chosen since no reattachment is expected for short orifices, the second one since reattachment was expected. Lastly, the third one represents the cross-flow cases. Due to lack of data in open literature, all the validation cases were made for sharp-edged orifices whereas the main focus of this study concerns radiused orifices. A summary can be seen in Table 5.17.

Table 5.17: Summary of validation cases.

<table>
<thead>
<tr>
<th>Case</th>
<th>L/D</th>
<th>r/D</th>
<th>Variable</th>
<th>Validation source</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0.4</td>
<td>0</td>
<td>( p_1/p_2 \in [1.01;1.7] )</td>
<td>Binder (2013)</td>
</tr>
<tr>
<td>2</td>
<td>1.5</td>
<td>0</td>
<td>( p_1/p_2 \in [1.01;1.4] )</td>
<td>Binder (2013)</td>
</tr>
<tr>
<td>3</td>
<td>2.0</td>
<td>0</td>
<td>( U/w_{ax} \in [0.17;0.74] )</td>
<td>Hüning (2012)</td>
</tr>
</tbody>
</table>

Validation case 1 was chosen not only due to the fact that it concerns a short orifice, but also due to the fact that Binder (2013) found an interesting behavior for this geometry. In the paper by Binder et al. (2014) which is based on the work of Binder (2013), a phenomena is discussed where the author noticed a sudden rise of the discharge coefficient usually together with a high frequency sound for a range of pressure ratios between 1.1 to 1.75 for \( L/D \) between 0.33 to 0.75. This trend was named bubble after its shape in the plots for the discharge coefficient as a function of the pressure ratio. For \( L/D = 0.33 \) and \( L/D = 0.4 \), the discharge coefficient returns to the same trend seen before the bubble when going beyond the bubble region. It is theorized that reattachment is not the underlying mechanism since the trend of the discharge coefficient would be altered outside the bubble region and not be the same before and after the bubble. Binder et al. (2014) speculate that the high frequency sound heard indicates that resonance might be present in the boundary layer in the orifice which could generate to oscillations of the vena contracta size. This would then yield a higher discharge coefficient since the time averaged size of the vena contracta would always be larger than the cross-sectional area without oscillations.

The results for case 1 can be seen in Figure 5.22. Firstly, the CFD calculation underpredicts the discharge coefficient slightly. The bubble phenomena can be seen for pressure ratios between 1.3 to 1.65 where the linear increase of the discharge coefficient is not present as seen below and above this range of pressure ratios. The CFD calculations do not capture the bubble region well; the discharge coefficient can be seen to increase in a linear manner without the slightest hint of assuming a bubble shape. Disregarding the region of the bubble, the trend of the CFD data follows quite well the experimental trend with a largest relative difference of 1.9% which strengthens the belief in the accuracy of future CFD calculations where the flow does not reattach.

Before leaving this case, it might be interesting to look at the separation behavior inside the orifice for a pressure ratio of 1.3 and 1.65 since the data is coherent between CFD and experimental and surround the bubble region. In Figure 5.23 (a) and (b), contour plots of the negative axial velocities are utilized to illustrate the separation region. From this, it can be seen that there for both pressure ratios, the flow stays separated and does not reattach. Thus,
the statement of Binder et al. (2014) this bubble phenomena should not be a result of reattachment can be seen to be at least correct for this geometry.

Validation case 2, with $L/D = 1.5$, $r/D = 0$ was also compared with experimental data from Binder (2013), and the results are presented in Figure 5.24. It can be seen that trend between CFD data and experimental data agrees quite well with CFD calculations again underpredicting the discharge coefficient. The largest relative difference was found to be 2.8%. In the mesh verification in subchapter 5.6.1.1, it could be seen that the flow is reattached for this geometry.
From these first two validation cases, it can be concluded that the CFD calculations give quite reasonable results for the discharge coefficient for flow cases without inlet cross-flow and more confidence can be put into future CFD results.

**Figure 5.24: Validation case 2. L/D = 1.5, r/D = 0.**

Validation case 3 is based on an experimental study by Hüning (2012). Due to time constraints, this validation was made with a limited amount of data points, as compared to previous validation cases. Please note that discharge coefficient used is the total discharge coefficient $C_{D,t}$ together with the cross-flow ratio based on the ideal velocity based on total pressure ratio $w_{ax}$. The static pressure ratio across the orifice was set to 1.05 in order to compare with Hüning (2012) whereas the main work regarding cross-flow has a slightly higher ratio of 1.10. It can be seen in Figure 5.25 that the CFD calculations once again underpredict the discharge coefficient. However, the trend agrees between experimental studies and CFD calculations although there is a constant discrepancy of $\Delta C_{D,t} \approx 0.03$. For higher values of $C_{D,t}$, this yields a lower relative difference but for lower values of $C_{D,t}$, the relative difference can amount to 10%. The experimental study had a quite high uncertainty of around 4% for this pressure ratio which can be seen in the same figure as error bars.

The conclusion of validation case 3 is that even though there is a systematic difference between experimental data and CFD data, the discharge coefficient trends are coherent between them. Thus future cross-flow studies through the use of CFD can be taken as valid. However, a CFD simulation is as the name implies just a simulation and all data should be validated experimentally before implementing into real-life applications.
Figure 5.25: Validation case 3, $L/D = 2.0$, $r/D = 0$, $p_1/p_2$. 
6 Results – Study I, no cross-flow

In this chapter the results are presented for Study I and the effects of different parameters will be examined and compared with literature. Comparison will also be made with the work of Binder (2013) and with existing correlations. A summary of the cases presented in the study can be found in Table 6.1.

Table 6.1: Summary of flow cases in Study I.

<table>
<thead>
<tr>
<th>L/D [-]</th>
<th>r/D [-]</th>
<th>p₁/p₂ [-]</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.4</td>
<td>0-0.20</td>
<td>1.05-1.4</td>
</tr>
<tr>
<td>2.0</td>
<td>0-0.30</td>
<td>1.05-1.4</td>
</tr>
<tr>
<td>4.0</td>
<td>0-0.30</td>
<td>1.05-1.4</td>
</tr>
</tbody>
</table>

Firstly, it can be observed that the highest pressure ratio has been limited to 1.4 due to the fact that supersonic regions where found for a pressure ratio of 1.5 as seen in subchapter 5.6.1.1. The lowest pressure ratio was set to 1.05 which is higher than the lowest pressure ratio in the validation studies in subchapter 5.7.2. The Reynolds number, based on the orifice diameter \( D \) and the isentropic velocity, is for all the flow cases larger than \( 10^4 \) which yields a Reynolds number independency as discussed in subchapter 3.2.3.2.

6.1 General overview

All the data are grouped after length to diameter ratio \( L/D \) and plotted as a function of pressure ratio \( p₁/p₂ \) in Figure 6.1 (a) and (b), and in Figure 6.2 (a). The discharge coefficient is shown to have a pressure dependency for all geometries to a certain degree. An augmentation of the radius to diameter ratio \( r/D \) can be seen to increase the discharge coefficient up to a certain limit where the rate of growth of the discharge coefficient declines. It can even be seen that too large radiusing decreases the discharge coefficient for short orifices with a length to diameter ratio \( L/D \) equal to 0.4. The effect of the length to diameter ratio \( L/D \) can be examined by regarding Figure 6.2 (b). It is observed that an elongation of the orifice is beneficial up the limit of \( L/D = 2.0 \), after which the discharge coefficient decreases. Finally, the orifices with \( L/D = 4.0 \) is observed to have quite similar behaviour to orifices with \( L/D = 2.0 \) when comparing Figure 6.1 (b) and Figure 6.2 (a).

In following subchapters a detailed analysis, as opposed to the mere observations in the current subchapter, will be carried out with focus on how the parameters and their interaction affect the discharge coefficient.
Figure 6.1: (a) Discharge coefficient $C_D$ as a function of pressure ratio $p_1/p_2$, $L/D = 0.4$. (b) Discharge coefficient $C_D$ as a function of pressure ratio $p_1/p_2$, $L/D = 2.0$. 

Figure 6.2: (a) Discharge coefficient $C_D$ as a function of pressure ratio $p_1/p_2$, $L/D = 4.0$. (b) Discharge coefficient $C_D$ as a function of length to diameter ratio $L/D$, $r/D = 0.04$. 
6.2 Detailed analysis

Three variables that affect the discharge coefficient were studied and this subchapter seeks to provide a detailed explanation of how the discharge coefficient depends on the variables both individually and combined.

6.2.1 L/D and p₁/p₂

A representative radius to diameter ratio \( r/D = 0.04 \) is chosen. Figure 6.3 (a) shows that an increase in pressure ratio \( p_1/p_2 \) increases the discharge coefficient for all length to diameter ratios \( L/D \). This is the result of the area increase of the vena contracta as separated region is suppressed which is in line with the discussion by Ward-Smith (1979). This can clearly be seen by comparing the height of the separation region by measuring the cross orifice distance of negative axial velocities. At the axial distance \( x/L = 0.5 \) for a length to diameter ratio \( L/D = 0.4 \) and radius to diameter ratio \( r/D = 0.04 \), the height is 0.033\(D \) for the case with pressure ratio \( p_1/p_2 = 1.05 \), but 0.026\(D \) for the case with a pressure ratio of 1.4. It can also be observed that the pressure ratio dependency is stronger for \( L/D = 0.4 \) compared with \( L/D = 2 \) and \( L/D = 4 \) which agrees with the findings of Deckker & Chang (1965). This might be illustrated by Figure 6.4 (a) and (b) where it can be observed that the recirculation region covers the whole orifice length for \( L/D = 0.4 \) and different pressure ratios presumably determines strongly the shape of the recirculation region and thus the pressure recovery possible.

Secondly, it can be observed in Figure 6.3 (b) that the discharge coefficient \( C_D \) increases as the length to diameter ratio \( L/D \) increases, but up to a certain limit. The discharge coefficient then decreases as \( L/D \) is increased. The increase of the discharge coefficient is due to the fact that reattachment (and thus pressure recovery) is present for \( L/D = 2.0 \) but not for \( L/D = 0.4 \) (true for small radiusing) and can be seen in Figure 6.4. A decrease of the discharge coefficient is seen between \( L/D = 2.0 \) and \( L/D = 4.0 \) and is presumed to be due to the frictional losses in the orifice since the trend between these two \( L/D \) are similar as evidenced in Figure 6.3 (a). For a pressure ratio \( p_1/p_2 = 1.4 \), the total pressure was mass flow averaged at two locations, the first located behind the recirculation region at the axial distance \( x/L = 0.5 \), and the second at the exit plane. The inlet conditions are included as reference. From Table 6.2, it is observed that the total pressure drop between the inlet and the region behind the recirculation region is 2.6% for \( L/D = 2.0 \), \( r/D = 0.04 \) and 2.22% for \( L/D = 4.0 \), \( r/D = 0.04 \). The pressure drop between the inlet and the exit plane is however 3.95% for the former case and 5.31% for the latter case. It can then be presumed that the larger total pressure drop is due to the frictional losses due to a longer orifice. This was also observed for the incompressible discharge coefficient by Lichtarowicz et al. (1965).

**Table 6.2: Pressure drop across the orifice.**

<table>
<thead>
<tr>
<th>L/D [-]</th>
<th>r/D [-]</th>
<th>Location</th>
<th>Total pressure [Pa]</th>
</tr>
</thead>
<tbody>
<tr>
<td>2.0</td>
<td>0.04</td>
<td>Inlet</td>
<td>142773</td>
</tr>
<tr>
<td></td>
<td></td>
<td>( x/L = 0.5 )</td>
<td>139023</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Exit plane</td>
<td>137138</td>
</tr>
<tr>
<td>4.0</td>
<td></td>
<td>Inlet</td>
<td>141782</td>
</tr>
<tr>
<td></td>
<td></td>
<td>( x/L = 0.5 )</td>
<td>138638</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Exit plane</td>
<td>135195</td>
</tr>
</tbody>
</table>
6.2.2 \( \frac{L}{D} \) and \( \frac{r}{D} \)

A representative pressure ratio \( \frac{p_1}{p_2} \) of 1.1 is chosen. It can be observed in the Figure 6.5 (a) that an increased radiusing augments the discharge coefficient, but the beneficial effects starts to decrease for high radius to diameter ratio \( \frac{r}{D} \) which agrees with the observations by Hay & Spencer (1992) for chamfered orifices. It is seen that the discharge coefficient even decreases when increasing \( \frac{r}{D} \) from 0.16 to 0.20 for \( \frac{L}{D} = 0.4 \). This can be explained by reattachment seen in Figure 6.6. The flow almost reattaches for \( \frac{r}{D} = 0.16 \) which increases the through flow. Conversely, the flow is separated at the exit of the orifice for \( \frac{r}{D} = 0.20 \) which

\[ \text{(a)} \]

Figure 6.3: (a) Discharge coefficient \( C_D \) versus the pressure ratio \( \frac{p_1}{p_2} \), \( \frac{r}{D} = 0.04 \). (b) Discharge coefficient \( C_D \) versus the length to diameter ratio \( \frac{L}{D} \), \( \frac{r}{D} = 0.04 \).

\[ \text{(b)} \]

\[ \text{(a)} \]

Figure 6.4: Recirculation region illustrated by contour plots of the negative axial velocity. (a) \( \frac{L}{D} = 0.4, \frac{r}{D} = 0.04, \frac{p_1}{p_2} = 1.4 \). (b) \( \frac{L}{D} = 2.0, \frac{r}{D} = 0.04, \frac{p_1}{p_2} = 1.4 \).
decreases the through flow. As a result of the large radiusing, the flow has a shorter distance to reattach. This behaviour has been observed previously by Hay & Spencer (1992), Dittman, et al. (2003) and Binder, et al. (2014) for chamfered orifices. The effective length $L_{eff}$ as proposed by Binder, et al. (2014) can thus be extended to radiused orifices as:

$\textit{L}_{\textit{eff},r} = \frac{L - r}{D} > 0.24$  \hspace{1cm} (6.1)

The effect of the length to diameter ratio $L/D$ has been discussed in the previous subchapter but an interesting effect can be noticed in Figure 6.5 (b), the discharge coefficient $C_D$ does not decrease between the length to diameter ratio $L/D = 2.0$ and $L/D = 4.0$ for sharp edged orifices with $r/D = 0$, the reason for this is unknown.

![Figure 6.5](image1.png)

(a) Discharge coefficient $C_D$ versus the radius to diameter ratio $r/D$, $p_1/p_2 = 1.1$. (b) Discharge coefficient $C_D$ versus the length to diameter ratio $L/D$, $p_1/p_2 = 1.1$.

![Figure 6.6](image2.png)

(a) Recirculation region illustrated by contour plots of the negative axial velocities. (a) $L/D = 0.4$, $r/D = 0.16$, $p_1/p_2 = 1.1$. (b) $L/D = 0.4$, $r/D = 0.20$, $p_1/p_2 = 1.1$. 

Figure 6.6: Recirculation region illustrated by contour plots of the negative axial velocities. (a) $L/D = 0.4$, $r/D = 0.16$, $p_1/p_2 = 1.1$. (b) $L/D = 0.4$, $r/D = 0.20$, $p_1/p_2 = 1.1$. 

Figure 6.5 (b) Discharge coefficient $C_D$ versus the radius to diameter ratio $r/D$, $p_1/p_2 = 1.1$. (b) Discharge coefficient $C_D$ versus the length to diameter ratio $L/D$, $p_1/p_2 = 1.1$. 

Figure 6.6: Recirculation region illustrated by contour plots of the negative axial velocities. (a) $L/D = 0.4$, $r/D = 0.16$, $p_1/p_2 = 1.1$. (b) $L/D = 0.4$, $r/D = 0.20$, $p_1/p_2 = 1.1$. 

Figure 6.6: Recirculation region illustrated by contour plots of the negative axial velocities.
6.2.3 \( r/D \) and \( p_1/p_2 \)

A representative \( L/D = 2.0 \) is chosen for this analysis. It can be noted that Figure 6.7 (a) is identical to Figure 6.1 (b). Virtually no difference in the discharge coefficient can be found between a radius to diameter ratio \( r/D = 0.16 \) compared with \( r/D = 0.30 \) as observed in Figure 6.8, the size is small compared to the same flow conditions but for \( L/D = 2.0 \) and \( r/D = 0.04 \) seen in Figure 6.4 (b). Thus, the orifice is more similar to a convergent nozzle for \( r/D = 0.16 \) and increasing the radius to diameter ratio \( r/D \) only increases the discharge coefficient marginally. Çengel & Cimbala (2006) defines a well-rounded inlet with negligible losses having a radius to diameter ratio \( r/D = 0.20 \) which is in line with the current observations. Figure 6.7 (b) reveals that the increase in pressure ratio \( p_1/p_2 \) only yields a slight raise in the discharge coefficient for all radius to diameter ratios for this specific \( L/D \).

\[ \text{Figure 6.7 (a) Discharge coefficient } C_D \text{ versus the pressure ratio } p_1/p_2, \text{ } L/D = 2.0. \text{ (b) Discharge coefficient } C_D \text{ versus the radius over diameter ratio } r/D, \text{ } L/D = 2.0. \]
Figure 6.8: Recirculation region illustrated by a contour plot of negative axial velocities. \( L/D = 2.0, r/D = 0.16, p_1/p_2 = 1.4 \).

6.2.4 L/D, r/D and \( p_1/p_2 \)

There are certain effects that are only present when the three parameters are varied. It has been discussed previously that for short orifices, \( L/D = 0.4 \), excessive radiusing is counterproductive. However, for high pressure ratios, the decrease in the discharge coefficient is not as pronounced as for lower pressure ratios as observed in Figure 6.9. This can be explained by once again regarding the separation regions. The recirculation regions for \( L/D = 0.4, p_1/p_2 = 1.4, r/D = 0.16 \) and \( r/D = 0.20 \) are shown in Figure 6.10 and it can be seen that due to the high pressure ratio, the flow reattaches for both cases which enables the discharge coefficient to remain fairly constant, something which is not the case in for low pressure ratios as seen previously in Figure 6.6.

Figure 6.9: Discharge coefficient \( C_D \) versus radius to diameter ratio \( r/D, L/D = 0.4 \).
Figure 6.10: Recirculation region illustrated by contour plots of negative axial velocities. (a) \( L/D = 0.4, p_1/p_2 = 1.4, r/D = 0.16 \). (b) \( L/D = 0.4, p_1/p_2 = 1.4, r/D = 0.20 \).

The discharge coefficient for short orifices with a length of diameter \( L/D = 0.4 \) are usually lower than for its longer counterparts. However, for high pressure ratios \( p_1/p_2 = 1.4 \) with a generous radius to diameter ratio \( r/D = 0.16 \), the discharge coefficient is higher for \( L/D = 0.4 \) than for \( L/D = 2.0 \) and \( L/D = 4.0 \) as evidenced from Figure 6.11. This is due to the fact that the flow is reattached even for \( L/D = 0.4 \) as seen in Figure 6.10 (a) and pressure recovery is possible. Thus, longer orifices yield a slightly lower discharge coefficient due to frictional losses.

Figure 6.11: Discharge coefficient \( C_D \) versus pressure ratio \( p_1/p_2, r/D = 0.16 \).

6.2.5 Additional findings

Even though the pressure ratio \( p_1/p_2 \) was limited to 1.4 in order to avoid supersonic regions, this was encountered for three flow cases listed in Table 6.3. However, the trends of the discharge coefficients as seen in Figure 6.1 (b) and Figure 6.2 (a) are consistent and thus these flow cases were included.
Table 6.3: Flow cases with supersonic regions.

<table>
<thead>
<tr>
<th>L/D [-]</th>
<th>r/D [-]</th>
<th>p₁/p₂ [-]</th>
</tr>
</thead>
<tbody>
<tr>
<td>2.0</td>
<td>0.04</td>
<td>1.4</td>
</tr>
<tr>
<td>2.0</td>
<td>0.08</td>
<td>1.4</td>
</tr>
<tr>
<td>4.0</td>
<td>0.08</td>
<td>1.4</td>
</tr>
</tbody>
</table>

The unsteady behaviour of short orifices have been mentioned previously and can be seen clearly in Figure 6.12 where contour plots of the negative axial velocities for L/D = 0.4, r/D = 0.16, p₁/p₂ = 1.05 are shown. Figure 6.12 (a) shows the instantaneous axial velocity at 0.01s and the Figure 6.12 (b) the time-averaged axial velocity. Visual inspection reveals that there is a considerable size difference which indicates that a transient behaviour is present.

Figure 6.12: Comparison of unsteadiness of the recirculation region, L/D = 0.4, r/D = 0.16, p₁/p₂ = 1.05. (a) Instantaneous axial velocity at 0.01 s. (b) Time-averaged axial velocity.
6.3 **Comparison with chamfered orifices**

Chamfered and radiused orifices both lead to smaller separation regions, but how do they compare against each other? Since this study is a continuation of the work by Binder (2013), the range of variables was intentionally kept similar to enable comparison. Binder (2013) studied chamfered orifices for length to diameter ratios $L/D$ of 0.4 and 2.14 which can directly be compared with the current study.

![Figure 6.13: Comparison between chamfered and radiused orifices, $L/D = 0.4$.](image)

From Figure 6.13, where orifices with a length over diameter ratio of $L/D = 0.4$ are presented, it can be observed that excessive chamfering leads to a lowered discharge coefficient as discussed by Binder, et al. (2014). It is seen to occur already at $w/D = 0.0276$ whereas the corresponding value for radiusing was found to be $r/D = 0.16$. This explains the higher $I_{eff}$ value recommended by Binder, et al. (2014). Moreover, it is apparent that small chamfering is preferable over small radiusing. A chamfer to diameter ratio $w/D = 0.022$ yields the approximately the same discharge coefficients as a radius to diameter ratio $r/D = 0.04$. Even more striking, the discharge coefficient associated with a chamfer to diameter ratio $w/D = 0.0276$ is even higher than the discharge coefficient for a radiused orifice with a radius to diameter ratio $r/D = 0.08$. Hay & Spencer (1992) found that for $L/D = 0.25$ and $p_1/p_2 = 1.2$, chamfering was more beneficial than radiusing up to a chamfer to diameter ratio $w/D = 0.08$ where the radiused orifices yielded higher discharge coefficients. This was not found for cases where $L/D = 0.5$ where chamfering performed slightly better than radiused orifices for all $w/D$ and $r/D$. The trend seen indicates that the orifices in Figure 6.13 behave similarly to the orifices studied by Hay & Spencer (1992). Still, no definite conclusion can be made since the largest chamfer to diameter studied by Binder (2013) was $w/D = 0.0531$.

In Figure 6.14 a comparison between chamfered (Binder, 2013) and radiused orifices is presented for $L/D = 2.0$-2.14. It is observed that the adverse effect of excessive chamfering is
not present here which is also the case with radiused orifices as discussed previously. It can be also observed that the benefits of increasing the chamfering are questionable for \( w/D > 0.0439 \), as higher chamfer to diameter ratios yield almost the same discharge coefficient. This is also found to be the case for radius to diameter ratios \( r/D \) larger than 0.16 as discussed previously. It is noticed that a radius to diameter ratio \( r/D = 0.08 \) yield a higher discharge coefficient than \( w/D = 0.152 \).

\[ \text{Figure 6.14: Comparison between chamfered and radiused orifices, } L/D = 2.0-2.14. \]

### 6.4 Comparison with correlations in literature

This subchapter will treat the comparison of the obtained data with existing correlations from open literature. Three correlations are included: the Parker and Kercher correlation (Parker & Kercher, 1991), the Idris and Pullen correlation (Idris & Pullen, 2005) and the Hüning correlation (Hüning, 2008). In a review (Hüning, 2008), the Parker and Kercher correlation is stated to be based on the incompressible discharge coefficient correlation presented by McGreehan & Schotsch (1988) connected with “empirical curve fits” with compressible test data “inter alia from Deckker & Chang (1965)” . The Hüning correlation from the same article uses a similar approach to the Parker and Kercher correlation with the difference being the treatment of the inlet radiusing and addition of cross-flow effects. The Idris and Pullen correlation is also based on the correlation from McGreehan & Schotsch (1988) and the compressibility correction comes as an extra term.

For \( L/D = 0.4 \) and \( r/D = 0.04 \), the high discharge coefficient from the current study is not predicted by any of the correlations as observed in Figure 6.15 (a). The Parker and Kercher correlation and the Hüning correlation best follow the trend by with a mean relative difference
of 8.1% and 5.5% respectively. The Idris and Pullen correlation has the same agreement in trend even though the slope of the discharge coefficient is not as similar and has a mean relative difference of 7.6%. For $L/D = 2.0$ shows that the Parker and Kercher, Hüning and Idris and Pullen correlations follow the trend and have a mean relative difference of 3.3%, 3.1% and 3.4% respectively. The Parker and Kercher correlation and subsequently the Hüning correlation are not valid for $L/D = 4.0$ and thus the comparison for $L/D = 4.0$ and $r/D = 0.04$ is made for the Idris and Pullen correlation seen in Figure 6.16. The Idris and Pullen correlation has a mean relative difference of 3.4%.

![Figure 6.15: Comparison of data with correlations. (a) $L/D = 0.4$, $r/D = 0.04$. (b) $L/D = 2.0$, $r/D = 0.04$.](image)
A summary of the characteristics of the agreement of the correlations with all the data is presented below:

- **L/D = 0.4**: Good agreement in trend between correlations for sharp edged orifices. The correlations yield higher discharge coefficients than the ones obtained. For radiused orifices, all correlations yield lower discharge coefficients compared with the calculated ones, however the trend is in agreement with the current study.

- **L/D = 2.0**: Good trend agreement is seen between all correlations and CFD results for sharp-edged orifices. For radiused orifices, all correlations present a lower discharge coefficient for low r/D and yield a higher discharge coefficient for high r/D.

- **L/D = 4.0**: The remaining correlation has the same behaviour as seen for L/D = 2.0.

The mean and maximum relative difference for all data points are presented in Table 6.4. It can be concluded that amongst the correlation, the Hüning correlation is most in agreement with this study, however the Idris and Pullen correlation which is also valid for L/D = 4.0 also yields good results. The Parker and Kercher correlation in which the Hüning was improved upon performed slightly worse than its successor.

*Table 6.4: Maximum and mean relative difference of correlations.*

<table>
<thead>
<tr>
<th>Correlation</th>
<th>Mean relative difference [%]</th>
<th>Max relative difference [%]</th>
</tr>
</thead>
<tbody>
<tr>
<td>Parker and Kercher (1991)</td>
<td>4.7</td>
<td>14.3</td>
</tr>
<tr>
<td>Hüning (2008)</td>
<td>4.0</td>
<td>11.8</td>
</tr>
<tr>
<td>Idris and Pullen (2005)</td>
<td>3.9</td>
<td>15.4</td>
</tr>
</tbody>
</table>
7 Results – Study II, cross-flow

In this chapter, the results from the cross-flow study, Study II, are presented and the effect of each parameter is presented. Moreover, comparisons will be made with previous studies with inlet cross-flow, axially rotating orifices and with existing correlations. A summary of the parameters is presented in Table 7.1.

Table 7.1: Summary of flow cases in Study II.

<table>
<thead>
<tr>
<th>L/D [-]</th>
<th>r/D [-]</th>
<th>U/c_{ax} [-]</th>
<th>p_{1}/p_{2} [-]</th>
</tr>
</thead>
<tbody>
<tr>
<td>2.0</td>
<td>0-0.30</td>
<td>0.17-1.12</td>
<td>1.1</td>
</tr>
<tr>
<td>4.0</td>
<td>0-0.30</td>
<td>0.17-1.12</td>
<td>1.1</td>
</tr>
</tbody>
</table>

The cross-flow ratio can be seen to range from moderate values up to very strong cross-flow where the cross-flow velocity exceeds the ideal orifice velocity. The software in use for secondary air system design employs static values for calculation so the cross-flow velocity ratio is based on the ideal orifice velocity based on the static pressure ratio $c_{ax}$ and the analysis will be made with a bias towards the static discharge coefficient $C_{D,s}$. The static pressure ratio was set to 1.1 which is a common operating range for the secondary air systems in gas turbines.
### 7.1 General overview

All the results are shown for the static discharge coefficient $C_{D,s}$ and the total discharge coefficient $C_{D,t}$ in Figure 7.1 and Figure 7.2 respectively. It should be reminded that they both stem from the same calculations but are just two different ways of calculating the discharge coefficient.

It can be observed that increasing the cross-flow ratio decreases the total discharge coefficient $C_{D,t}$ for all cases, but the static discharge coefficient $C_{D,s}$ for the highest radius to diameter ratios $r/D$ increases and can exceed unity. For a length to diameter ratio $L/D = 2.0$, the static discharge coefficient decreases after the increase seen by the radius to diameter ratio $r/D = 0.3$ but its counterpart for $L/D = 4.0$ increases with increasing cross-flow ratio with no decrease in the range of this study. For the same $L/D$, the static discharge coefficient for $r/D = 0.20$ increases and then decreases with increasing cross-flow ratio. It is also observed that increasing the radius to diameter ratio $r/D$ increases both the static and total discharge coefficients. Elongating the orifice from $L/D = 2.0$ to $L/D = 4.0$ can be seen to be beneficial to the static and total discharge coefficient. In the next subchapter, a more detailed analysis will be carried out.

![Graphs showing discharge coefficients](a) and (b)

*Figure 7.1: Static discharge coefficient $C_{D,s}$ (a) $L/D = 2.0$. (b) $L/D = 4.0*
7.2 Detailed analysis

This subchapter seeks out to provide a thorough explanation of the effects of the variables and their combined effects on the static discharge coefficient $C_{D,s}$.

7.2.1 L/D and $U/c_{ax}$

A representative radius to diameter ratio $r/D = 0.08$ is chosen for the analysis. It can be observed from Figure 7.3 (a) that the static discharge coefficient is reduced with increasing cross-flow ratio. $L/D = 4.0$ yields a higher static discharge coefficient than $L/D = 2.0$. Figure 7.3 (b) shows that for low cross-flow ratios, there seem to be no difference in static discharge coefficient between $L/D = 2.0$ and $L/D = 4.0$ but for high cross-flow ratios, there is certainly a benefit with increasing $L/D$. Figure 7.4 illustrates the separation region for $L/D = 2.0$, $r/D = 0.08$, $U/c_{ax} = 0.689$ and $L/D = 4.0$, $r/D = 0.08$, $U/c_{ax} = 0.689$. The separation region is certainly larger than for cases without cross-flow and furthermore asymmetric. The larger separation region reduces the through flow (Hünig, 2008) but it is observed that the flow reattaches for $L/D = 4.0$. The reattachment is most likely the cause of the improvement of the discharge coefficient.
Figure 7.3 (a) Static discharge coefficient $C_{D,s}$ versus the cross-flow ratio $U/c_{ax}$, $r/D = 0.08$. (b) Static discharge coefficient $C_{D,s}$ versus the length to diameter ratio $L/D$, $r/D = 0.08$.

Figure 7.4: Recirculation region illustrated by contour plots of negative axial velocities. (a) $L/D = 2.0$, $r/D = 0.08$, $U/c_{ax} = 0.689$. (b) $L/D = 4.0$, $r/D = 0.08$, $U/c_{ax} = 0.689$.

7.2.2 $L/D$ and $r/D$

A representative cross flow ratio $U/c_{ax} = 0.259$ is chosen. Figure 7.5 (a) reveals that similarly to the case with no cross flow, the benefits of radiusing is not as pronounced for large radiusing. Furthermore, Figure 7.5 (b) reveals that there is a small benefit for orifices with small or no radiusing to increase the length to diameter ratio $L/D$. 
Figure 7.5: (a) Static discharge coefficient $C_{D,s}$ versus the radius to diameter ratio $r/D$, $U/c_{ax} = 0.259$. Right: Static discharge coefficient $C_{D,s}$ versus the length to diameter ratio $L/D$, $U/c_{ax} = 0.259$.

7.2.3 $r/D$ and $U/c_{ax}$

A length to diameter $L/D = 4.0$ is chosen for this analysis. Firstly, Figure 7.6 (a) is identical to Figure 7.1 (b). The increase in the static discharge coefficient $C_{D,s}$ with increasing cross-flow ratio $U/c_{ax}$ for the highest radius to diameter $r/D = 0.30$ is quite remarkable. It should also be noted that the static discharge coefficient exceeds unity, i.e. the mass flow exceeds the ideal mass flow. The mechanism behind this is the fact that there exists a stagnation point at the orifice inlet as seen in Figure 7.7 (a). Due to the stagnation region, there is a rise in static pressure as evidenced in Figure 7.7 (b). The static pressure is now higher (shown as gauge pressure in the figure) and will yield a higher pressure ratio than the original static pressure ratio. The flow is thus driven by a pressure ratio higher than the original static pressure equivalent to 1.1. Figure 7.8 presents the mass-flow averaged total pressure at the orifice axial distance $x/L = 0.75$ and at the exit plane $x/L = 1$ divided with the ambient static pressure. It can be seen that the pressure ratio is indeed larger than 1.1, i.e. some of the dynamic pressure from the cross-flow has been redirected into the orifice. This is in line with the discussion in subchapter 3.2.3.3 (Hüning, 2008). Figure 7.6 (b) illustrates that the benefits for increasing radius to diameter ratio $r/D$ are most pronounced for strong cross flow ratios.
Figure 7.6: (a) Static discharge coefficient $C_{D,s}$ versus the cross flow ratio $U/c_{ax}$, $L/D = 4.0$. (b) Static discharge coefficient $C_{D,s}$ versus the radius to diameter ratio $r/D$, $L/D = 4.0$.

Figure 7.7: Stagnation region, $L/D = 4.0$, $r/D = 0.30$, $U/c_{ax} = 1.12$. (a) Velocity contour plot. (b) Static pressure contour plot.
7.2.4 \( \text{L/D, r/D and } U/c_{ax} \)

Some features only manifest themselves when all three parameters are varied. Figure 7.1 shows that the static discharge coefficient increase seen for \( \text{L/D} = 2.0, \text{r/D} = 0.30 \) is followed by a decrease where as that is not the case for \( \text{L/D} = 4.0, \text{r/D} = 0.30 \). It is believed that increasing the cross-flow ratio sufficiently high, the static discharge coefficient will start to decrease. It is also discovered that for strong cross-flow ratios of \( U/c_{ax} = 1.12 \), increasing the length to diameter ratio \( \text{L/D} \) is beneficial for all \( \text{r/D} \) as illustrated by Figure 7.9. This was not the case for weaker cross-flow ratios as seen in Figure 7.5 (b).

Figure 7.8: Pressure ratio \( p_{\text{torifice}}/p_2 \) at \( x/L = 0.75 \) and \( x/L = 1.0 \) for \( \text{L/D} = 4.0, \text{r/D} = 0.30 \).

Figure 7.9: Static discharge coefficient \( C_{D,s} \) versus the length to diameter ratio \( \text{L/D, U/c}_{ax} = 1.12 \)
7.3 Comparison with data from inlet cross-flow and rotating orifices

There is a lack of data in open literature regarding cross-flow with radiused orifices. Rohde, et al. (1969) presented two orifices with length to diameter ratio $L/D = 1.06$, radius to diameter ratio $r/D = 0.1951$ and $r/D = 0.4878$ respectively and it is compared with $L/D = 2.0$, $r/D = 0.20$ and $r/D = 0.30$ in Figure 7.10 (a). Additionally, a comparison can be made with $L/D = 4.0$, $r/D = 0$ and is presented in Figure 7.10 (b). There are quite many differences in how Rohde, et al. (1969) present the data. To start with, they utilize a velocity head ratio $\Theta_v$ defined as:

$$\Theta_v = \frac{p_{1,t} - p_{2,s}}{p_{1,t} - p_{1,s}}$$  \hspace{1cm} (7.1)

A transformation of the velocity head ratio to the cross-flow velocity ratio utilized in this work can be found in the article by Hüning (2008) and is based on the ideal velocity $w_{ax}$ based on the total upstream pressure. Furthermore the data is presented for a constant main duct Mach number and not constant static pressure ratio which is the case for the current work. Hüning (2008) also provides a transformation law for the static pressure ratio across the orifice and it is calculated that the static pressure ratio varies between 1.01 up to 1.6 whereas the current study only has a static pressure ratio of 1.1. Nonetheless, the trend of a decreasing discharge coefficient with increasing cross-flow ratio is consistent between Rohde, et al. (1969) and the current study even no direct comparison can be made in Figure 7.10 (a) due to the length to diameter ratio disparity. Conversely, in Figure 7.10, it can be observed that the CFD data underpredict the discharge coefficient, a trend which has been observed in the validation study with data from Hüning (2012).

*Figure 7.10: Comparison with Rohde, et al. (1969). (a) $L/D = 1.06-2.0$, $r/D = 0.1951-0.4878$. (b) $L/D = 4.0$, $r/D = 0$.)*
Jakoby, et al. (1997) studied axially rotating orifices and utilized both the absolute and relative definition of the discharge coefficient, $C_{D,abs}$ and $C_{D,rel}$ respectively. The configuration was similar to the current study with a pressure ratio $p_1/p_2 = 1.1$, $L/D = 2.66$ and $r/D = 0-0.5$. A comparison can be made but one has to keep in mind that axially rotating orifices and inlet cross-flow are not identical as discussed in subchapter 3.2.3.4. Figure 7.11 presents all four definitions of the discharge coefficients where the absolute discharge coefficient $C_{D,abs}$ can be compared with the static discharge coefficient $C_{D,s}$ and the relative discharge coefficient $C_{D,rel}$ can be compared with the total discharge coefficient $C_{D,t}$. In Figure 7.11 (a), the trends of the current study and the work of Jakoby, et al. (1997) agree to some extent, with decreasing discharge coefficients with increasing cross-flow ratio. However, the current study yields consistently lower discharge coefficients and the small rise in the absolute discharge coefficient is not present. In Figure 7.11 (b), Jakoby, et al. (1997) present an increase followed by a decrease of the absolute discharge coefficient with increasing cross-flow ratio. This is also seen in the current study where the static discharge coefficient increases, exceeds unity and then decreases with increasing cross-flow ratio. However, the maximum is shifted to higher cross-flow ratios for this case. They also experience a rising absolute discharge coefficient for $L/D = 2.66$, $r/D = 0.5$ not followed by a decrease with increasing cross-flow ratio. This is similar to the findings of this study for $L/D = 4.0$, $r/D = 0.3$ as seen in the Figure 7.6 (a).

![Figure 7.11: Comparison with Jakoby, et al. (1997). (a) $L/D = 2.0-2.66$, $r/D = 0.08-0.1$. (b) $L/D = 2.0-2.66$, $r/D = 0.30$.](image)

Dittman, et al. (2003) studied experimentally axially rotating short orifices with the largest length to diameter ratio of $L/D = 1.25$. The relative discharge coefficient was found to be independent of pressure ratio for the largest $L/D = 1.25$ and largest $r/D = 0.5$ and the data presented here does not have a specific pressure ratio $p_1/p_2$ but is rather an average of all pressure ratios as understood by Hüning (2008). Dittman, et al. (2003) use the cross-flow ratio defined by the ideal velocity $w_{ax,rel}$ based on the relative total upstream values as presented in Equation (3.15), comparable with the ideal velocity $w_{ax}$ seen in the current study. Furthermore, the relative discharge coefficient $C_{D,rel}$ is presented which should be compared
with the total discharge coefficient $C_{D,t}$ in the current study. As in the previous case, no direct comparison can be made between the current study and the work by Dittman, et al. (2003) due to the additional effect experienced by rotating orifices and also due to the shorter length of the orifice. Nonetheless, it is observed in Figure 7.12 that trends agree quite well with the discharge coefficient decreasing with increasing cross-flow ratio. The current study does not experience such a sharp decrease for low cross-flow ratios as present in the rotating case. Furthermore, the discharge coefficient for the current study can be observed to be elevated compared with the rotating orifices in most cases.

![Figure 7.12: Comparison with Dittman et al. (2003), L/D = 1.25-2.0, r/D = 0-0.20.](image)

### 7.4 Comparison with correlations in literature

There is only one correlation from the previous chapter that include the effect of inlet cross-flow; the Idris and Pullen correlation (the Hüning correlation has a highest $L/D = 1.25$ for inlet cross-flow and is not applicable for the current cases).

Idris and Pullen (2005) included the effect of rotation in their correlation based on experimental studies. While an explicit upper limit of the cross-flow ratio is not stated in their correlation, the highest cross-flow ratio $U/c_{ax}$ presented from the experimental work is 0.53. An extrapolation is suggested for larger values which is not included in this comparison. This makes this correlation quite limited for comparison since the cross-flow ratio for the current study extends to 1.12. However, Figure 7.13 shows that the trend from the correlation match quite well with the current data even though the correlation is based on data from rotating orifices.
Figure 7.13: Comparison with Idris and Pullen correlation. (a) L/D = 2.0, r/D = 0-0.30. (b) L/D = 4.0, r/D = 0-0.30.

The maximum and mean relative difference can be seen in Table 7.2. From the current analysis, it is concluded that the Idris and Pullen correlation works well but is quite limited in range.

**Table 7.2: Max and mean relative difference of correlations**

<table>
<thead>
<tr>
<th>Correlation</th>
<th>Mean relative difference [%]</th>
<th>Max relative difference [%]</th>
</tr>
</thead>
<tbody>
<tr>
<td>Idris and Pullen</td>
<td>3.2</td>
<td>6.6</td>
</tr>
</tbody>
</table>
The ultimate goal with this thesis is to implement the data into a secondary air system program which solves large complex flow networks. The data obtained in this study can be used in its current form with interpolation for variable values in between the ones studied. It is however more convenient to implement a correlation. Three different correlations were created, the first one belonging to the case without cross-flow, Study I. The second and third correlations belong to Study II and concern the static and total discharge coefficient. The current secondary air system software uses only one of the two definitions of the discharge coefficient but it is useful to present correlations for both for future purposes.

8.1 Correlation- Study I

There are multiple ways to correlate data, and this work utilizes a relatively simple approach. The ultimate goal is to correlate the data to provide a discharge coefficient as a function dependent on the three studied variables, i.e. $C_D(L/D, r/D, p_1/p_2)$. The correlation is based on all the data obtained in chapter 6, including the flow cases with a small supersonic region and the range is presented in Table 6.1. The procedure to create the correlation can be described in three steps:

1. Create a second degree polynomial fit for each geometry as a function of the pressure ratio $p_1/p_2$. The result is 17 different correlations $C_D(L/D, r/D, p_1/p_2)$ with three coefficients for each correlation.

2. Group the correlation into three groups dependent on the length to diameter ratio $L/D$. Correlate the constants in the previously obtained correlations with each other with a fourth degree polynomial as a function of the radius to diameter ratio $r/D$. Three correlations $C_D(L, r, p_1/p_2)$, one for each $L/D$ are obtained. Each correlation has 15 coefficients.

3. Correlate the coefficients in the three correlations with each other with a second degree polynomial as a function of the length to diameter ratio $L/D$ and obtain a final correlation for the discharge coefficient $C_D(L/D, r/D, p_1/p_2)$ with 45 coefficients.

The correlation can be written in a compact form, with all coefficients stored in the third order tensor $a_{ijk}$, as:

$$C_D(L/D, r/D, p_1/p_2) = \sum_{i=0}^{2} \sum_{j=0}^{4} \sum_{k=0}^{2} a_{ijk} \left( \frac{L}{D} \right)^i \left( \frac{r}{D} \right)^j \left( \frac{p_1}{p_2} \right)^k$$

The correlation is, in spite of its complex form, simple to implement in a software. The coefficients can be found in Table C. 1. The performance of the correlation is excellent with a maximum error compared with the data it is based on of 1.87% and a mean error of 0.3%. In Figure 8.1, the discharge coefficients given by the correlation are plotted against the discharge coefficients the correlation is based on. The points should ideally be located on the line but it
can be seen that for high discharge coefficients, the correlation tend to yield slightly higher values compared to the discharge coefficients on which it is based on.

Figure 8.1: Discharge coefficients given by the correlation versus the discharge coefficients from the current study.

The correlation also yields good values for intermediate values of $L/D$, $r/D$ and $p_1/p_2$ in almost all cases. An example can be seen in Figure 8.2 (a) where the discharge coefficient line for $r/D = 0.02$ provided by the correlation is located between the discharge coefficients lines for $r/D = 0$ and $r/D = 0.04$. The discharge coefficient line of $r/D = 0.06$, which is given by the correlation, is found between the lines from $r/D = 0.04$ and $r/D = 0.08$. There is however one region where the correlation perform poorly. For values of $L/D = 2.0-4.0$ and $r/D = 0.30$, the correlation is good for $L/D = 2.0$ and $L/D = 4.0$ but for $L/D$ values in between where there are no data points, the correlation present an unphysical behavior. Figure 8.2 (b) reveals that for $L/D = 2.7$ and $L/D = 3.4$, the discharge coefficient is even lower than the discharge coefficient for $L/D = 4.0$ and the line has a curved shape which is not present in the CFD data. As a matter of fact, this problem extends down to $r/D = 0.25$. The suggestion is to limit the general correlation to a highest radius to diameter ratio $r/D = 0.2$. This is then in agreement with the shortest orifice with a length to diameter ratio $L/D = 0.4$ which did not include $r/D = 0.30$. If a radius to diameter ratios $0.20 < r/D \leq 0.30$ are to be investigated, it is proposed that the discharge coefficient for $L/D = 2.0$ and $L/D = 4.0$ are calculated for the desired radius to diameter ratio $r/D$. Subsequently a linear interpolation can be utilized to find the discharge coefficient for the desired $L/D$. 
8.2 Correlations-Study II

Two sets of data were produced from the cross-flow study, one for the static discharge coefficient $C_{D,s}$ and one for the total discharge coefficient $C_{D,t}$. The cross-flow ratio is based on the static pressure ratio and is thus $U/c_{ax}$. The creations of the correlations are based on the same procedure presented in subchapter 8.1 and based on the data from chapter 7, in total 34 data points. The steps are identical for the total and static discharge coefficients and are outlined for a general discharge coefficient $C_{D,x}$ below:

1. Create a correlation for each geometry with a third degree polynomial as a function of $U/c_{ax}$. This yields 8 correlations for the discharge coefficient $C_{D,x}(\frac{U}{c_{ax}})$ with four coefficients each.

2. Sort the correlations into two different groups as a function of $L/D$ and correlate the coefficients of the correlations in respective group with a third order polynomial as a function of $r/D$. Two discharge coefficient correlations $C_{D,x}(\frac{U}{c_{ax}}, \frac{r}{L})$ with 16 coefficients are created from this step.

3. Correlate the coefficients in the two correlations with each other with a first degree polynomial as a function of $L/D$, this yields a final correlation for the discharge coefficient $C_{D,x}(\frac{U}{c_{ax}}, \frac{r}{L}, \frac{B}{D})$ with 32 coefficients.

The correlations for the static discharge coefficient $C_{D,s}$ and the total discharge coefficient $C_{D,t}$ can be written in a compact form as presented in Equation (8.2) and Equation (8.3) with the coefficients stored in the third order tensor $b_{ijk}$ and $c_{ijk}$ respectively. The coefficients can be found in Table C.2 and Table C.3.
\[ C_{D,s} \left( \frac{U}{C_{ax}}, \frac{r}{D}, \frac{L}{D} \right) = \sum_{i=0}^{3} \sum_{j=0}^{3} \sum_{k=0}^{1} b_{ijk} \left( \frac{L}{D} \right)^k \left( \frac{r}{D} \right)^j \left( \frac{U}{C_{ax}} \right)^i \]  
\[ (8.2) \]

\[ C_{D,t} \left( \frac{U}{C_{ax}}, \frac{r}{D}, \frac{L}{D} \right) = \sum_{i=0}^{3} \sum_{j=0}^{3} \sum_{k=0}^{1} c_{ijk} \left( \frac{L}{D} \right)^k \left( \frac{r}{D} \right)^j \left( \frac{U}{C_{ax}} \right)^i \]  
\[ (8.3) \]

The performance of the two correlations are excellent with Equation (8.2) having a maximum error of 3.27% and a mean error of 0.45% and Equation (8.3) having a maximum error of 1.07% and mean error of 0.13%. The static discharge coefficients from the correlation are plotted against the static discharge coefficients they are based on in Figure 8.3. The correlation is optimistic and yield slightly higher values for the values not on the line. Figure 8.4 shows that the total discharge coefficients given by the correlation agree quite well with the data since most of the points are located on the line. Furthermore, the correlations work well for all intermediate variable values, e.g. Figure 8.5 shows that for intermediate values of \( r/D \), the correlation presents reasonable values.

![Figure 8.3: Static discharge coefficients \( C_{D,s} \) given by the correlation versus the static discharge coefficients \( C_{D,s} \) from the current study.](image)
Figure 8.4: Total discharge coefficients $C_{D,t}$ given by the correlation versus the total discharge coefficients $C_{D,t}$ from the current study.

Figure 8.5: Comparison of intermediate variable values with CFD data. $L/D = 2.0$, $r/D = 0.08-0.30$
(a): Static discharge coefficient $C_{D,s}$ (b): Total discharge coefficient $C_{D,t}$

8.3 Concluding remarks

An overview of the correlations and its properties can be found in Table 8.1.
Table 8.1: Overview of correlations.

<table>
<thead>
<tr>
<th>Correlated variable</th>
<th>L/D</th>
<th>r/D</th>
<th>p₁/p₂</th>
<th>U/cₐₓ</th>
</tr>
</thead>
<tbody>
<tr>
<td>Cₐₓ</td>
<td>0.4-4.0</td>
<td>0-0.30</td>
<td>1.05-1.4</td>
<td>0</td>
</tr>
<tr>
<td>Cₐₓₕ</td>
<td>2.0-4.0</td>
<td>0-0.30</td>
<td>1.1</td>
<td>0.17-1.12</td>
</tr>
<tr>
<td>Cₐₓ₉</td>
<td>2.0-4.0</td>
<td>0-0.30</td>
<td>1.1</td>
<td>0.17-1.12</td>
</tr>
</tbody>
</table>

The first correlation for Cₐₓ without cross-flow has to be used with care for 0.20 < r/D ≤ 0.30 due to several reasons. Firstly, it is not defined for L/D < 2.0 since no calculations were made for L/D = 0.4. Secondly, it provides unphysical results in the range 0.20 < r/D ≤ 0.30 if L/D ≠ 2.0 or 4.0. A possible remedy has been discussed in subchapter 8.1. The correlations for cross-flow can be used everywhere inside the range of the variables.

All the correlations are based on polynomial functions which are meaningful inside the range of correlated data. However, if the correlations are used outside of the specified range, the results will not be physically meaningful or correct due to the polynomial functions. Therefore, great care must be taken to not to use the correlations outside their range or employ a method to deal with out of bound variables. The simplest way is to use the highest value allowed for the variable out of bounds. More sophisticated methods include the use of damping functions based on observations by other authors to achieve a physical meaningful behaviour outside the range. None of these methods have been implemented in the current work and are presented as possible future improvements. Some small suggestions based on the observations of the current study are however given here:

- If the radius to diameter ratio r/D exceeds 0.20, the discharge coefficient will decrease for L/D = 0.4. For L/D ≥ 2.0, the discharge coefficient can be taken as constant for r/D > 0.2.
- For L/D > 4.0, the discharge coefficient can be taken to linearly decrease as seen for 2.0 ≤ L/D ≤ 4.0.
- For p₁/p₂ > 1.4, the discharge coefficient is dependent on the choking behavior which is dependent on both L/D and r/D. Binder (2013) provides an excellent overview of choking behavior for different L/D and w/D which should be similar to the current study.

Correlations are very convenient and present an elegant way to join data points, however the usefulness of the correlations can be questioned in this work. For the correlation for the discharge coefficient without cross-flow Cₐₓ, 85 data points are used and the final correlation has 45 coefficients. It might perhaps be simpler to include all the data points and use interpolation to find the desired discharge coefficients. Furthermore, this would avoid the problems experienced with the correlation in chapter 8.1. This is more evident when regarding the correlations for the discharge coefficients related to the cross-flow study. The correlations have 32 coefficients whereas the data points only amount to 34. Thus very little has been gained by implementing a correlation in terms of stored values.
# 9 Conclusion

This work is a continuation of the experimental work by Binder (2013) conducted at Siemens Industrial Turbomachinery, Finspång Sweden. The objective was to, through CFD, investigate additional parameters, both geometrical and physical, regarding flow through orifices commonly used in secondary air systems in gas turbines.

The first goal was to investigate the effect of radiused orifices in combination with different orifice length to diameter and pressure ratios. A verification of the computational model was performed to ensure that the mesh was sufficient for the needs of the study and also a study of turbulence models was conducted and compared against experimental data. To validate the model, comparisons were made with experimental data by Binder (2013) with good agreement, although the observed “bubble” effect could not be replicated. The flow behaviour of sharp and radiused orifices were examined and deeper insight of the flow behaviour was gained. The results were compared with data from chamfered orifices by Binder (2013) and existing correlations and in general, good agreement was found. A correlation based on polynomials was created and was found to agree well with the data it is based on although there exists some limitations.

The second goal was to investigate the behaviour of orifices with an inlet cross-flow. No mesh verification was made due to time constraints but a validation study was made based on data from Hüning (2012). An analysis of the flow behaviour was conducted with the most interesting find being the fact that dynamic pressure from the cross-flow is recovered in the orifice giving a rise of the static pressure and in some cases an increase of the orifice through flow. Due to the lack of data for inlet cross-flow on radiused orifices, comparisons were made mainly with studies and correlations concerning axially rotating orifices and similar trends could be observed. Two correlations were created for the two definitions of the discharge coefficient presented in the current study and the performance of the correlations is remarkably good in the range.

The study for inlet cross-flow is quite limited due to the time frame of this work. A natural continuation point would be to study orifices with $L/D = 0.4$ and $r/D = 0-0.30$ since the meshes already exist and requires quite little work to setup. Moreover, intermediate cross-flow ratios should be run for existing geometries since the current study only had four to five cross-flow ratios per geometry. Furthermore, different pressure ratios $p_1/p_2$ could also be studied. A mesh verification study should also be performed for representative cases to ensure the mesh is sufficient for the task at hand, something that unfortunately was not performed in the current work.

Finally, CFD simulations should be validated by experiments if possible and it is recommended to do so especially for short orifices ($L/D = 0.4$) since the values obtained are relatively high compared to existing correlations.
Bibliography


## Appendix A

Table A. 1: Overview of the meshes used in chapters 6 and 7.

<table>
<thead>
<tr>
<th>L/D</th>
<th>r/D</th>
<th>Hemisphere I elements</th>
<th>Hemisphere II elements</th>
<th>Inner geometry elements [-]</th>
<th>Cross-flow cube elements [-]</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.4</td>
<td>0</td>
<td>66305</td>
<td>175986</td>
<td>1210464</td>
<td>-</td>
</tr>
<tr>
<td>0.4</td>
<td>0.04</td>
<td>-</td>
<td>163976</td>
<td>910848</td>
<td>-</td>
</tr>
<tr>
<td>0.4</td>
<td>0.08</td>
<td>-</td>
<td>163976</td>
<td>1069824</td>
<td>-</td>
</tr>
<tr>
<td>0.4</td>
<td>0.16</td>
<td>-</td>
<td>163976</td>
<td>1140480</td>
<td>-</td>
</tr>
<tr>
<td>0.4</td>
<td>0.20</td>
<td>-</td>
<td>163976</td>
<td>1193472</td>
<td>-</td>
</tr>
<tr>
<td>2.0</td>
<td>0</td>
<td>105872</td>
<td>281879</td>
<td>1315008</td>
<td>-</td>
</tr>
<tr>
<td>2.0</td>
<td>0</td>
<td>-</td>
<td>953756</td>
<td>4914784</td>
<td>-</td>
</tr>
<tr>
<td>2.0</td>
<td>0.04</td>
<td>-</td>
<td>163976</td>
<td>1259712</td>
<td>-</td>
</tr>
<tr>
<td>2.0</td>
<td>0.08</td>
<td>-</td>
<td>163976</td>
<td>1325952</td>
<td>-</td>
</tr>
<tr>
<td>2.0</td>
<td>0.08</td>
<td>-</td>
<td>953756</td>
<td>1325952</td>
<td>190894</td>
</tr>
<tr>
<td>2.0</td>
<td>0.16</td>
<td>-</td>
<td>163976</td>
<td>1387776</td>
<td>-</td>
</tr>
<tr>
<td>2.0</td>
<td>0.20</td>
<td>-</td>
<td>163976</td>
<td>1414272</td>
<td>-</td>
</tr>
<tr>
<td>2.0</td>
<td>0.20</td>
<td>-</td>
<td>953756</td>
<td>1414272</td>
<td>190894</td>
</tr>
<tr>
<td>2.0</td>
<td>0.30</td>
<td>-</td>
<td>163976</td>
<td>1555584</td>
<td>-</td>
</tr>
<tr>
<td>2.0</td>
<td>0.30</td>
<td>-</td>
<td>953756</td>
<td>1555584</td>
<td>190894</td>
</tr>
<tr>
<td>4.0</td>
<td>0</td>
<td>105872</td>
<td>281879</td>
<td>1685664</td>
<td>-</td>
</tr>
<tr>
<td>4.0</td>
<td>0</td>
<td>-</td>
<td>953756</td>
<td>6011464</td>
<td>-</td>
</tr>
<tr>
<td>4.0</td>
<td>0.04</td>
<td>-</td>
<td>163976</td>
<td>1250880</td>
<td>-</td>
</tr>
<tr>
<td>4.0</td>
<td>0.08</td>
<td>-</td>
<td>163976</td>
<td>1325952</td>
<td>-</td>
</tr>
<tr>
<td>4.0</td>
<td>0.08</td>
<td>-</td>
<td>953756</td>
<td>1325952</td>
<td>190894</td>
</tr>
<tr>
<td>4.0</td>
<td>0.16</td>
<td>-</td>
<td>163976</td>
<td>1467264</td>
<td>-</td>
</tr>
<tr>
<td>4.0</td>
<td>0.20</td>
<td>-</td>
<td>163976</td>
<td>1612992</td>
<td>-</td>
</tr>
<tr>
<td>4.0</td>
<td>0.20</td>
<td>-</td>
<td>953756</td>
<td>1612992</td>
<td>190894</td>
</tr>
<tr>
<td>4.0</td>
<td>0.30</td>
<td>-</td>
<td>163976</td>
<td>1749888</td>
<td>-</td>
</tr>
<tr>
<td>4.0</td>
<td>0.30</td>
<td>-</td>
<td>953756</td>
<td>1749888</td>
<td>190894</td>
</tr>
</tbody>
</table>
Appendix B

Figure B. 1: Axial velocity profile across the orifice at $x/L = 0.476$. $L/D = 1.5$, $r/D = 0$.

Figure B. 2: Turbulent kinetic energy profile across the orifice at $x/L = 0.476$. $L/D = 1.5$, $r/D = 0$. 
Figure B. 3: Axial velocity profile across the orifice at $x/L = 0.6$. $L/D = 0.4$, $r/D = 0.04$.

Figure B. 4: Turbulent kinetic energy profile across the orifice at $x/L = 0.6$. $L/D = 0.4$, $r/D = 0.04$. 
Figure B. 5: Axial velocity profile across the orifice at $x/L = 0.8$, $L/D = 0.4$, $r/D = 0.04$.

Figure B. 6: Turbulent kinetic energy profile across the orifice at $x/L = 0.8$, $L/D = 0.4$, $r/D = 0.04$. 
## Appendix C

### Table C.1: Coefficients $a_{ijk}$.

<table>
<thead>
<tr>
<th>$i = 0$</th>
<th>$k = 0$</th>
<th>$k = 1$</th>
<th>$k = 2$</th>
</tr>
</thead>
<tbody>
<tr>
<td>$j = 0$</td>
<td>0.377</td>
<td>0.3023</td>
<td>-0.05472</td>
</tr>
<tr>
<td>$j = 1$</td>
<td>1.848</td>
<td>-2.295</td>
<td>0.3039</td>
</tr>
<tr>
<td>$j = 2$</td>
<td>-126.6</td>
<td>89.97</td>
<td>-14.02</td>
</tr>
<tr>
<td>$j = 3$</td>
<td>-660.2</td>
<td>568.9</td>
<td>-99.44</td>
</tr>
<tr>
<td>$j = 4$</td>
<td>5391</td>
<td>-4204</td>
<td>707.8</td>
</tr>
<tr>
<td>$i = 1$</td>
<td>$k = 0$</td>
<td>$k = 1$</td>
<td>$k = 2$</td>
</tr>
<tr>
<td>$j = 0$</td>
<td>0.2052</td>
<td>-0.1729</td>
<td>0.03631</td>
</tr>
<tr>
<td>$j = 1$</td>
<td>-0.3961</td>
<td>5.785</td>
<td>-0.9304</td>
</tr>
<tr>
<td>$j = 2$</td>
<td>276.4</td>
<td>-228.5</td>
<td>37.28</td>
</tr>
<tr>
<td>$j = 3$</td>
<td>419.8</td>
<td>-339.2</td>
<td>61.26</td>
</tr>
<tr>
<td>$j = 4$</td>
<td>-7364</td>
<td>5670</td>
<td>-953</td>
</tr>
</tbody>
</table>

### Table C.2: Coefficients $b_{ijk}$.

<table>
<thead>
<tr>
<th>$i = 0$</th>
<th>$k = 0$</th>
<th>$k = 1$</th>
</tr>
</thead>
<tbody>
<tr>
<td>$j = 0$</td>
<td>0.7406</td>
<td>0.01075</td>
</tr>
<tr>
<td>$j = 1$</td>
<td>3.41</td>
<td>-0.2585</td>
</tr>
<tr>
<td>$j = 2$</td>
<td>-16.79</td>
<td>1.085</td>
</tr>
<tr>
<td>$j = 3$</td>
<td>27.64</td>
<td>-1.825</td>
</tr>
<tr>
<td>$i = 1$</td>
<td>$k = 0$</td>
<td>$k = 1$</td>
</tr>
<tr>
<td>$j = 0$</td>
<td>-0.1719</td>
<td>0.0151</td>
</tr>
<tr>
<td>$j = 1$</td>
<td>4.802</td>
<td>-1.302</td>
</tr>
<tr>
<td>$j = 2$</td>
<td>-25.1</td>
<td>8.747</td>
</tr>
<tr>
<td>$j = 3$</td>
<td>26.33</td>
<td>-12.42</td>
</tr>
<tr>
<td>$i = 2$</td>
<td>$k = 0$</td>
<td>$k = 1$</td>
</tr>
<tr>
<td>$j = 0$</td>
<td>-0.08821</td>
<td>0.02515</td>
</tr>
<tr>
<td>$j = 1$</td>
<td>-19.41</td>
<td>4.762</td>
</tr>
<tr>
<td>$j = 2$</td>
<td>157.7</td>
<td>-36.5</td>
</tr>
<tr>
<td>$j = 3$</td>
<td>-254.8</td>
<td>58.66</td>
</tr>
<tr>
<td>$i = 3$</td>
<td>$k = 0$</td>
<td>$k = 1$</td>
</tr>
<tr>
<td>$j = 0$</td>
<td>-0.06188</td>
<td>0.007975</td>
</tr>
<tr>
<td>$j = 1$</td>
<td>11.75</td>
<td>-2.985</td>
</tr>
<tr>
<td>$j = 2$</td>
<td>-114.5</td>
<td>26.74</td>
</tr>
<tr>
<td>$j = 3$</td>
<td>206.2</td>
<td>-46.58</td>
</tr>
</tbody>
</table>
Table C. 3: Coefficients $c_{ijk}$

<table>
<thead>
<tr>
<th>$i = 0$</th>
<th>$k = 0$</th>
<th>$k = 1$</th>
</tr>
</thead>
<tbody>
<tr>
<td>$j = 0$</td>
<td>0.742</td>
<td>0.01075</td>
</tr>
<tr>
<td>$j = 1$</td>
<td>3.662</td>
<td>-0.3215</td>
</tr>
<tr>
<td>$j = 2$</td>
<td>-19.15</td>
<td>1.63</td>
</tr>
<tr>
<td>$j = 3$</td>
<td>32.06</td>
<td>-2.815</td>
</tr>
<tr>
<td>$i = 1$</td>
<td>$k = 0$</td>
<td>$k = 1$</td>
</tr>
<tr>
<td>$j = 0$</td>
<td>-0.1821</td>
<td>0.0148</td>
</tr>
<tr>
<td>$j = 1$</td>
<td>2.583</td>
<td>-0.7468</td>
</tr>
<tr>
<td>$j = 2$</td>
<td>-2.904</td>
<td>3.699</td>
</tr>
<tr>
<td>$j = 3$</td>
<td>-15.7</td>
<td>-3.24</td>
</tr>
<tr>
<td>$i = 2$</td>
<td>$k = 0$</td>
<td>$k = 1$</td>
</tr>
<tr>
<td>$j = 0$</td>
<td>-0.4827</td>
<td>0.02135</td>
</tr>
<tr>
<td>$j = 1$</td>
<td>-16.2</td>
<td>3.62</td>
</tr>
<tr>
<td>$j = 2$</td>
<td>108.5</td>
<td>-24.34</td>
</tr>
<tr>
<td>$j = 3$</td>
<td>-154.6</td>
<td>35.59</td>
</tr>
<tr>
<td>$i = 3$</td>
<td>$k = 0$</td>
<td>$k = 1$</td>
</tr>
<tr>
<td>$j = 0$</td>
<td>0.2084</td>
<td>-0.0064</td>
</tr>
<tr>
<td>$j = 1$</td>
<td>10.25</td>
<td>-2.382</td>
</tr>
<tr>
<td>$j = 2$</td>
<td>-84.62</td>
<td>18.81</td>
</tr>
<tr>
<td>$j = 3$</td>
<td>140.3</td>
<td>-30.7</td>
</tr>
</tbody>
</table>